HEIDENHAIN

TNC7 basic

User's Manual
Complete Edition

NC Software
81762x-18

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

> **i** HEIDENHAIN offers separate information products for other target groups:
> - Leaflets and overview of the product portfolio 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 **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for running NC programs.
    ID:
  - 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:
  - The **Machining Cycles** User's Manual contains all functions of the machining cycles.
    ID:
  - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
    ID:
- As PDF files, sub-divided according to the printed versions or as a **Complete edition** User's Manual, containing all modules
  ID:
  TNCguide
- As an HTML file used as the **TNCguide** product aid integrated directly into 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 use", Page 73

Further information products for users

The following information products are available to you:

- **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 comprise 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 important 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 indicates a **cross reference**.
> A cross reference leads to external documentation for example the documentation of your machine manufacturer or other supplier.

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 aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.
Further information: "Available user documentation", Page 63

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended use", Page 73

Related topics

- The **Help** workspace
  Further information: "The Help workspace", Page 1158

Requirement

In the factory default setting, the control offers the integrated product aid **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** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired languageversion (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 product aid **TNCguide** can be selected within the **Help** application or in the **Help** workspace.
Further information: "The Help application", Page 67
Further information: "The Help workspace", Page 1158

Operation of **TNCguide** is identical in both cases.
Further information: "Icons", Page 68

The Help application
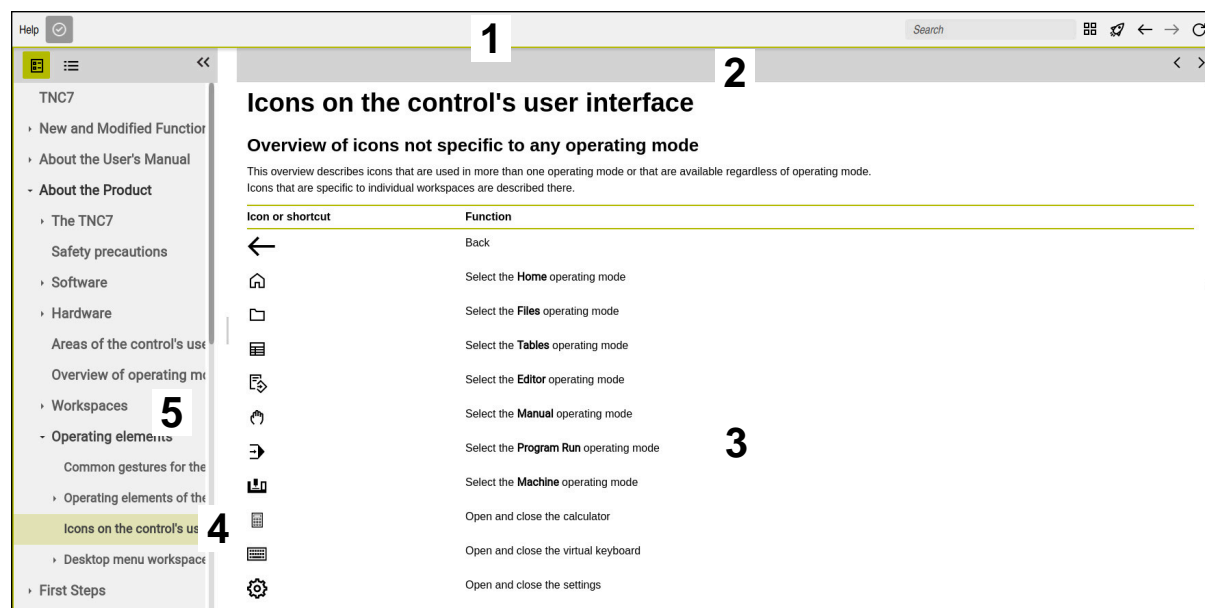
Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

1   Title bar of the **Help** workspace
    Further information: "The Help workspace", Page 68
2   Title bar of the integrated product aid **TNCguide**
    Further information: "TNCguide ", Page 68
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**

Icons

The Help workspace

The **Help** workspace within the **Help** application includes the following icons:

Icon	Meaning
⊘	Open or close the **Search results** column **Further information:** "Search in TNCguide", Page 69
▦	**Open Home page** The start page displays all available documentation. Select the desired documentation using 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.
⬈	**Open Tutorials**
← →	**Navigate** Navigate between the contents opened recently
⟳	**Refresh**

TNCguide

The integrated **TNCguide** product aid includes the following icons:

Icon	Meaning
▤	**Open Structure** The structure consists of the content headings. The structure serves for main navigation within the documentation.
☰	**Open Index** The index consists of important keywords. The index serves as an alternative navigation within the documentation.
‹ ›	**Navigate** Display previous or next page within the documentation
« »	**Open or close** Display or hide the navigation
▣	**Copy** Copy NC examples to the clipboard **Further information:** "Copying NC examples to clipboard", Page 70

Context-sensitive help

You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function).

To call context-sensitive help, the following elements are available:

Icon or key	Meaning
⊙	**Help** icon If you select the icon and then one of the items in the user interface, the control will open the associated information in **TNCguide**.
HELP	**HELP** key If you press the **HELP** key while editing an NC block, the control will display the associated information in **TNCguide**.

If you call TNCguide in a certain context, the control opens the contents in a pop-up window. If you select the **Show more** button, the control will open **TNCguide** in the **Help** application.

Further information: "The Help application", Page 67

If the **Help** workspace is already open, the control displays **TNCguide** there and will not open a pop-up window.

Further information: "The Help workspace", Page 1158

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 enter a character.
>
> 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
▶ Expand **Notes on using NC programs**
▶ Read and follow **Notes on using NC programs**
   Further information: "Notes on using NC programs", Page 65

▶ 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 content according to the **Notes on using NC programs**
▶ Use the Simulation mode to test the NC program
   Further information: "The Simulation Workspace", Page 1197

1.6 Contacting the editorial staff

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

We are continuously striving 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 basic

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 basic additionally offers you graphical or form-based programming so that you can attain the desired results with speed and reliability.

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

Operation is made easier, for example, by using touch probes, handwheels or a 3D mouse.

Further information: "Hardware enhancements", Page 89

Definitions

Abbreviation	Definition
TNC	**TNC** is derived from the acronym **CNC** (computerized numerical control). The **T** (tip or touch) stands for the capability of entering 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.
basic	The addition basic indicates that the control provides all required basic functions for universal milling or drilling.

2.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling 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.

> **i** 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, for example, in function adaptations of the controls and safety precautions in the information products.

> **i** Contribute actively to increasing the safety by reporting any missing or misleading information.
> **Further information:** "Contacting the editorial staff", Page 70

2.1.2 Intended place of operation

In accordance with the DIN EN 50370-1 standard for 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.

> ℹ️ 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

⚠ 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 manufacturer

---

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!

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 for 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 a **TOOL CALL 0**. There is a risk 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. If the previous value is kept, there is a danger of collision!

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

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

NOTICE

Caution: Data may be lost!

If you do not properly remove a connected USB device during a data transfer, then data may be damaged or deleted!

▶ Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs

▶ Use the **Eject** 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 regardless of the control's status!

▶ 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.

> **ℹ** The actual range of functions depends, among other things, on the enabled software options.
> **Further information:** "Software options", Page 78

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

> **ℹ** 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
817620-18	TNC7 basic
817625-18	TNC7 basic 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.
> If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

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 1786

The TNC7 basic features various software options that the machine manufacturer may enable separately, even at a later point in time. The following overview includes only those software options that are relevant for you.

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 basic can be equipped with a **SIK1** or **SIK2** plug-in board. Depending on which one is used, the numbers of the software options differ.

> **i** The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The parentheses enclose the **SIK1** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1).
>
> The Technical Manual informs about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

SIK2 option numbers are structured by <class>-<option>-<version>:

Class	The function is effective for the following areas:
	■ 1: Programming, simulation, and process setup
	■ 2: Part quality and productivity
	■ 3: Interfaces
	■ 4: Technology functions and quality assessment
	■ 5: Process stability and monitoring
	■ 6: Machine configuration
	■ 7: Developer tools
Option	Sequential number within each class
Version	New versions of software options are released if, for example, its features have been changed.

You can order some software options with **SIK2** more than once in order to obtain multiple variants of the same function (e.g., if you need to enable multiple control loops for the axes). In the User's Manual, these software option numbers are identified by an asterisk (*****).

The control indicates in the **SIK** menu item of the **Settings** application whether a software option has been enabled, and if so, how often.

Further information: "The SIK menu item", Page 1785

Overview

> ℹ️ Keep in mind that particular software options also require hardware extensions.
> **Further information:** "Hardware", Page 84

Software option	Definition and application
Control Loop Qty. (#0-3 / #6-01-1*)	**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, for example, for detachable and motor-driven tilting tables. If your control features a **SIK2**, you can order this software option multiple times and enable up to 8 control loops.

Adv. Function Set 1 (#8 / #1-01-1)	**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 742
	■ Programming of contours on a developed cylindrical surface (e.g., with Cycle **27 CYLINDER SURFACE)**
	  Further information: "Cycle 27 CYLINDER SURFACE (#8 / #1-01-1)", Page 916
	■ Programming the rotary axis feed rate in mm/min with **M116**
	  Further information: "Interpreting the feed rate for rotary axes in mm/min with M116 (#8 / #1-01-1)", Page 978
	■ 3-axis circular interpolation with a tilted working plane
	
	The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.

Adv. Function Set 2 (#9 / #4-01-1)	**Advanced functions (set 2)**
	
	On machines with rotary axes, this software option enables simultaneous 4-axis machining of workpieces.
	
	The software option includes the following functions:
	
	■ **TCPM** (tool center point management): Automatic tracking of linear axes during rotary axis positioning
	  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
	■ Running of NC programs with vectors, including optional 3D tool compensation
	  Further information: "3D tool compensation (#9 / #4-01-1)", Page 805
	■ Manual moving of axes in the active tool coordinate system **T-CS**

Touch Probe Function (#17 / #1-05-1)	**Touch-probe functions** This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.

Software option	Definition and application
	The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces. **Further information:** "Tastsystemfunktionen in der Betriebsart Manuell", Page
HEIDENHAIN DNC (#18 / #3-03-1)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Adv. Function Set 3 (#21 / #4-02-1)	**Advanced functions (set 3)** This software option offers additional ease of use with two powerful miscellaneous functions. The software option includes the following miscellaneous functions: ■ **M120** for machining small contour steps without error message and contour damage **Further information:** "Pre-calculating a radius-compensated contour with M120 (#21 / #4-02-1)", Page 981 ■ **M118** for superimposed handwheel positioning during program run **Further information:** "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979 The advanced functions (set 3) reduce the setup effort and increase flexibility during program run.
Collision Monitoring (#40 / #5-03-1)	**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 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 machine downtime. **Further information:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
CAD Import (#42 / #1-03-1)	**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.

Software option	Definition and application
	Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
Adaptive Feed Contr. (#45 / #2-31-1)	**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 machining times without adapting the NC program, while at the same time preventing machine damage from overload. **Further information:** "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884
KinematicsOpt (#48 / #2-01-1)	**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 for temperature-induced deviations through repeated measurements and corrections. **Further information:** "Touch-Probe Cycles for Kinematics Measuring", Page 1575
OPC UA NC Server Qty. (#56-61 / #3-02-1*)	**OPC UA NC Server** These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features a **SIK2**, you can order this software option multiple times and enable up to six connections. **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804
4 Additional Axes (#77 / #6-01-1*)	**Four additional control loops** **Further information:** "**Control Loop Qty.** (#0-3 / #6-01-1*)", Page 79

Ext. Tool Management (#93 / #2-03-1)	**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 (#93 / #2-03-1)", Page 1706
	 ■ The **T usage order** shows the tool order of the NC program or pallet to be run
	     Further information: "T usage order (#93 / #2-03-1)", Page 1704
	
	 Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.

Remote Desktop Manager (#133 / #3-01-1)	**Remote Desktop Manager** This software option is used to display and operate externally linked computer units.

Software option	Definition and application
	With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase the efficiency. **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

Collision Monitoring (#140 / #5-03-2)	**Dynamic Collision Monitoring DCM version 2**
	
	This software option includes all functions of the Dynamic Collision Monitoring DCM (#40 / #5-03-1) software option.
	
	In addition, this software option provides the following features:
	▪ Collision monitoring of fixtures
	  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859
	▪ Define reduced minimum distance between fixture and tool
	  Further information: "Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)", Page 878

Cross Talk Comp. (#141 / #2-20-1)	**Compensation of axis couplings (CTC)** Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adapt. Contr. (#142 / #2-21-1)	**Position Adaptive Control (PAC)** Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adapt. Contr. (#143 / #2-22-1)	**Load Adaptive Control (LAC)** Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adapt. Contr. (#144 / #2-23-1)	**Motion Adaptive Control (MAC)** Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Contr. (#145 / #2-30-1)	**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) (#145 / #2-30-1)", Page 893
Machine Vibr. Contr. (#146 / #2-24-1)	**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 (#152 / #1-04-1)	**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.

Software option	Definition and application
	Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124
Batch Process Mngr. (#154 / #2-05-1)	**Batch Process Manager (BPM)** This software option makes it easy to plan and execute multiple production jobs. By extending and combining the pallet management and extended tool management functions (#93 / #2-03-1), the 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:** "The Job list workspace", Page 1620
Component Monitoring (#155 / #5-02-1)	**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.
Model Aided Setup (#159 / #1-07-1)	**Graphically supported setup** This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model.
Opt. Contour Milling (#167 / #1-02-1)	**Optimized contour machining (OCM)** This software option enables 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 STANDARD FIGURES** for frequently needed contours With OCM you can shorten machining times while at the same time reducing tool wear. **Further information:** "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626

2.3.2 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

(os)ⓘ

- ▶ Double-tap or double-click **About HeROS**
- > The control opens the **HEROS Licence Viewer** window.

OPC UA

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

The control's behavior can be influenced by means of the OPC UA NC Server (#56-61 / #3-02-1*) and HEIDENHAIN DNC (#18 / #3-03-1). 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 (#56-61 / #3-02-1*)", Page 1804

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 77

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

2.4.1 Touchscreen and keyboard unit
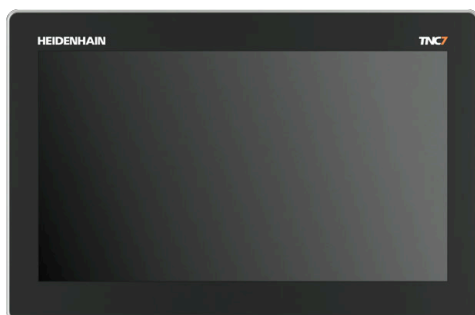
16" MC 345 with TE 340 (FS)

The TNC7 basic is delivered with a 16-inch screen.

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

Further information: "Common gestures for the touchscreen", Page 98

Further information: "Operating elements of the keyboard unit", Page 99

The machine operating panel is machine-dependent.

Operating and cleaning the touchscreen

Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. Small amounts of liquid do not affect the function of the touchscreen, but large amounts may cause incorrect input.

Switch off the control before cleaning the touchscreen. As an alternative, you can use the touchscreen cleaning mode.

Further information: "The Settings Application", Page 1777

Do not apply the cleaning agent directly to the screen, but slightly dampen a clean, lint-free cleaning cloth with it.

The following cleaning agents are permitted for the screen:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

The following cleaning agents are prohibited for the screen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

> ⓘ ■ Touchscreens are sensitive to electrostatic charges from the user. Dissipate the static charge by touching metallic, grounded objects or wear ESD clothing.
> ■ Wear operating gloves to prevent the screen from getting dirty.
> ■ You can operate the touchscreen with special touchscreen operating gloves.

Cleaning the keyboard unit

Switch the control off before cleaning the keyboard unit.

NOTICE
Caution: risk of property damage
Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.
▶ Use permitted cleaning agents only
▶ Use a clean, lint-free cleaning cloth to apply the cleaning agent

The following cleaners are permitted for the keyboard unit:
- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:
- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

> **ⓘ** Wear work gloves to prevent the keyboard unit from getting dirty.

If a trackball is embedded in the keyboard, you need to clean it only if it no longer works properly.

To clean a trackball (if needed):
- ▶ Shut down 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 the cleaning agent onto a cleaning cloth
- ▶ 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 2009

> ⚙ IP54 protection cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:
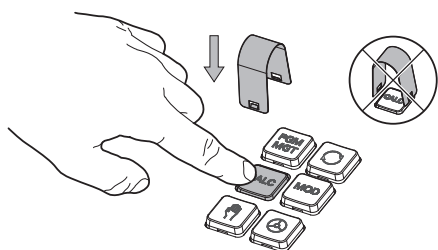
▶ Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

> ℹ Pressing the key will make it easier to apply the keycap puller.
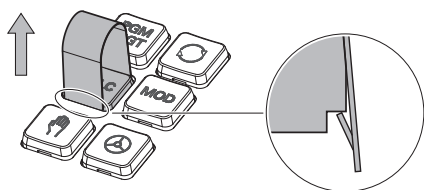
▶ Pull off the keycap
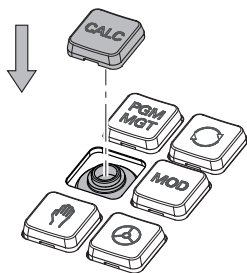
▶ Place the keycap onto the seal and push it down

> ℹ The seal must not be damaged; otherwise IP54 protection cannot be guaranteed.

▶ Verify proper seating and correct functionality

2.4.2 Hardware enhancements

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

The TNC7 basic features various hardware extensions that the machine manufacturer may add separately, even at a later point in time. The following overview includes only those extensions that are relevant to you.

> **i** Keep in mind that particular hardware enhancements require additional software options.
> **Further information:** "Software options", Page 78

Hardware enhancements	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 1741
Workpiece touch probes	With this extension, the control can determine locations on the workpiece and misalignments automatically and precisely (#17 / #1-05-1). 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 (#17 / #1-05-1)", Page 1257
Tool touch probes	With this extension, the control can measure tools automatically and precisely, directly in the machine (#17 / #1-05-1). 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. **Further information:** "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
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.

Hardware enhancements	Definition and application
	📖 **VTC User's Manual** All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID: 1322445-xx
Additional operating stations	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 860 is an auxiliary screen that increases the area of the main screen. This allows multiple applications to be viewed simultaneously. ℹ️ By adding a keyboard unit, the ITC 860 can be used as a full-fledged additional operating station. The additional operating stations increase operator comfort, especially on large machining centers.
Industrial PC	You use this enhancement to install and run Windows-based applications. With Remote Desktop Manager (#133 / #3-01-1), you can display applications on the control screen. **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819 The industrial PC is a secure and powerful alternative to external PCs.
Override controller	This extension allows you to define breakpoints at which the control stops duringprogram run (e.g., before a tilting function). The override controller enables the feed rate or rapid traverse value to be changed as well as starting or continuing the NC program. **Further information:** "Override Controller", Page 1756

2.5 Areas of the control's user interface
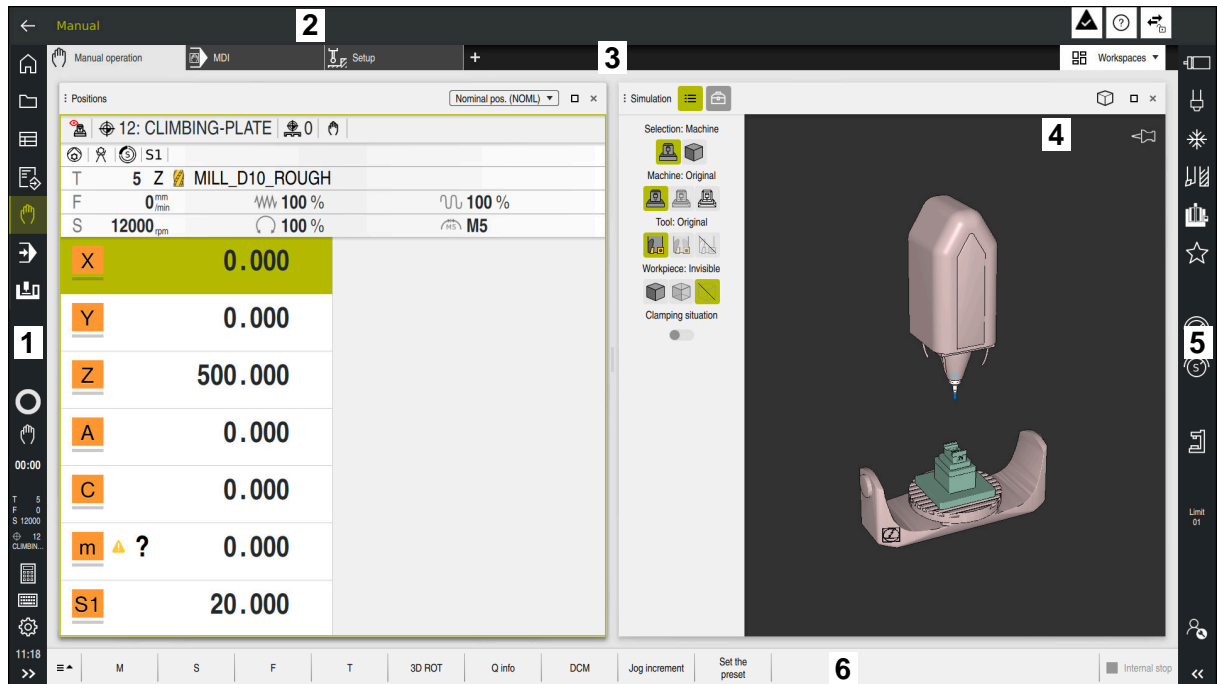
User interface of the control in the **Manual operation** application

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 the operating modes", Page 93
- Status overview
  Further information: "Status overview on the TNC bar", Page 155
- Calculator
  Further information: "Calculator", Page 1179
- Screen keyboard
  Further information: "Virtual keyboard of the control bar", Page 1160
- Settings
  The Settings menu enables you to change the control interface:
  - **Left-hand mode**
    The control swaps the positions of the TNC bar and the machine manufacturer bar.
  - **Dark Mode**
    In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether **Dark Mode** is available for selection.
  - **Font size**
- Date and time

2   Information bar
- Active operating mode
- Message menu
  Further information: "Message menu on the information bar", Page 1193

- **Help** icon for context-sensitive help

  Further information: "Context-sensitive help", Page 69
- Symbols

3 Application bar
- Tabs of opened applications

  The maximum number of simultaneously opened applications is limited to ten tabs. If you try to open an eleventh tab, the control shows a message.
- Selection menu for workspaces

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

4 Workspaces

Further information: "Workspaces", Page 94

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 the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
	The **Home** operating mode contains the following applications:	
	■ The **Start/Login** application	
	During the startup process, the control is in the **Start/Login** application.	
	■ The **Settings** application	Page 1777
	■ The **Help** application	Page 1158
	■ Applications for machine parameters	Page 1833
	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 822
	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 1666
	In the **Editor** operating mode you can do the following:	Page 202
	■ Create, edit and simulate NC programs	
	■ Create and edit contours	
	■ Create and edit pallet tables	
	The **Manual** operating mode contains the following applications:	
	■ The **Manual operation** application	Page 186
	■ The **MDI** Application	Page 1221
	■ The **Setup** application	Page 1257
	■ The **Move to ref. point** application	Page 181
	■ The **Retract** application	Page 1661
	You can move the tool away from the workpiece, for example after a power failure.	
	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.	Page 1638
	You also execute pallet tables in this operating mode.	
	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.	Page 1765
	Refer to your machine manual.	
	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
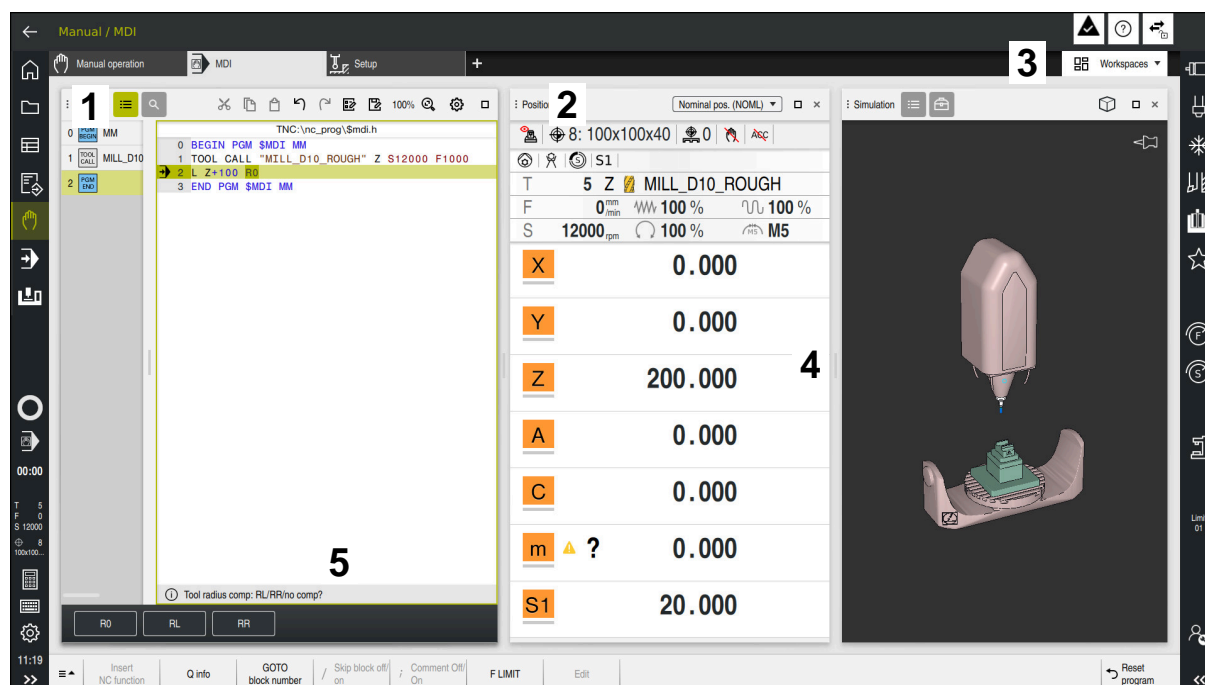
The control in the **MDI** application with three open workspaces

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:

Symbol	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 (#17 / #1-05-1) 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 1257
Job list In the **Job list** workspace, you edit and execute pallet tables.	Page 1620
Open File In the **Open File** workspace you select or create files, for example.	Page 834
Files 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 **Files** workspace is part of the **Files** operating mode.	Page 822
Details In the **Details** workspace, the control displays information on the selected machine parameter or the last change you made.	Page 1838
Document You can open files for viewing in the **Document** workspace, for example a technical drawing.	Page 837
Settings In the **Settings** workspace, you can display and edit, if required, various settings of the control (e.g., set up the traverse limits). The **Settings** workspace is part of the **Settings** application.	Page 1777
The **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 1676
The **Form** for pallets In the **Form** workspace the control shows the contents of the pallet table for the selected row.	Page 1628
Retract In the **Retract** workspace, you can disengage the tool after a power interruption.	Page 1661
Desktop menu In the **Desktop menu** workspace, the control displays selected control and HEROS functions.	Page 109
Help In the **Help** workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid **TNCguide**.	Page 1158
Contour graphics	Page 1089

Workspace	Further information
In the **Contour graphics** workspace, you can 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 graphics** workspace for graphical editing.	
List	Page 1835
In the **List** workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	
Positions	Page 149
In the **Positions** workspace, the control displays information about the status of various functions of the control and about current axis positions.	
Program	Page 203
The control displays the NC program in the **Program** workspace.	
Referencing	Page 181
On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.	
Remote Desktop Manager (#133 / #3-01-1)	Page 1765
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.	
Quick selection	Page 835
In the **Quick selection new table** and **Quick selection new file** workspaces, you can create files or open existing files, depending on the active operating mode.	
Simulation	Page 1197
In the **Simulation** workspace, the control shows the simulated or current movements, depending on the operating mode.	
Simulation status	Page 170
In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	
Start/Login	Page 113
In the **Start/Login** workspace, the control shows the steps that are performed during startup.	
Status	Page 157
In the **Status** workspace, the control shows the status and values of individual functions.	
Table	Page 1670
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.	
The **Table** for machine parameters	Page 1835
In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	

Workspace	Further information
Keyboard	Page 1160
In the **Keyboard** workspace, you can enter NC functions, letters and numbers, and also navigate.	
Overview	Page 1772
In the **Overview** workspace, the control displays information on the status of individual functional safety (FS) safety functions.	

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 basic primarily through the touchscreen, meaning with gestures.

Further information: "Common gestures for the touchscreen", Page 98

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.

> **i** If there are deviations from the virtual keyboard, the table also indicates the corresponding keys on the virtual keyboard.
> **Further information:** "Virtual keyboard of the control bar", Page 1160

Keycaps for alphabetic keyboard

Key	Meaning
A B C	Enter texts (e.g., file names)
Q Q QL QR	**Q** With an open NC program, enter a Q parameter formula in the **Editor** operating mode, or in the **Manual** operating mode open the **Q parameter list** window **Further information:** "The Q parameter list window", Page 1010 By selecting the **Q** key multiple times, you can switch between **Q**, **QL**, and **QR**.
ESC	Close windows and context menus
⇄	Select the next element; for example, an input field, button, or selection option
SHIFT + TAB	Select the previous element
PRT SC	Create screenshot
⟨⟩	The **DIADUR** keys provide the following functions: ■ Left **DIADUR** key Open the **HEROS menu** ■ Right **DIADUR** key Open the **Remote Desktop Manager** connection in the defined desktop **Further information:** "Connection settings", Page 1821
▤	Open the context menu in the **Klartext editor** or in the text editor

Keycaps for operating aids

Key	Meaning
PGM MGT	Open the **Open File** workspace in the **Editor** and **Program Run** operating modes **Further information:** "The Open File workspace", Page 834
◯	Currently no function
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 1193
CALC	Open and close the calculator **Further information:** "Calculator", Page 1179
MOD	Open the **Settings** application **Further information:** "The Settings Application", Page 1777
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 66

Operating modes

ℹ️ On the TNC7 basic 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, for example, instead activate a toggle switch.

Key	Meaning
🖐	Open the **Manual operation** application in the **Manual** operating mode **Further information:** "The Manual operation application", Page 186
⊙	Activate and deactivate the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic Handwheel", Page 1741
⇥	Open the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 269
▢	Open the **MDI** application in the **Manual** operating mode **Further information:** "The MDI Application ", Page 1221
▣	Open the **Program Run** operating mode in **Single Block** mode **Further information:** "The Program Run operating mode", Page 1638
➡	Open the **Program Run** operating mode **Further information:** "The Program Run operating mode", Page 1638
⇕	Open the **Editor** operating mode **Further information:** "The Editor operating mode", Page 202
➡	While the NC program is running, open the **Simulation** workspace in the **Editor** operating mode **Further information:** "The Simulation Workspace", Page 1197

Keycaps for NC dialog

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

Key	Meaning
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function **Further information:** "Fundamentals of approach and departure functions", Page 330
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode **Further information:** "Graphical programming", Page 1089
CHF	Program a chamfer **Further information:** "Chamfer CHF", Page 302
L	Program a straight line segment **Further information:** "Straight line L", Page 300
CR	Program a circular arc with radius entry **Further information:** "Circular path CR", Page 310
RND	Program a rounding arc **Further information:** "Rounding RND", Page 304
CT	Program a circular arc with tangential connection to the preceding contour element **Further information:** "Circular path CT", Page 313
CC	Program a circle center or pole **Further information:** "Circle center point CC", Page 306
C	Program a circular arc with reference to the circle center **Further information:** "Circular path C ", Page 308
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
CYCL DEF	In the **Insert NC function** window, open the **Fixed cycles** folder in order to select a cycle **Further information:** "Defining cycles", Page 224
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 227
LBL SET	Program a jump label **Further information:** "Defining a label with LBL SET", Page 360
LBL CALL	Program a subprogram or a program section repeat **Further information:** "Calling a label with CALL LBL", Page 361
STOP	Program an intentional stop

Key	Meaning
	Further information: "Programming the STOP function", Page 964
TOOL DEF	Pre-select a tool in the NC program
	Further information: "Tool pre-selection by TOOL DEF", Page 286
TOOL CALL	Call the tool data in the NC program
	Further information: "Tool call by TOOL CALL", Page 279
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)
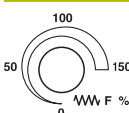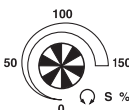
Keycaps for axis input and value input

Key	Meaning
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 **Further information:** "Status overview on the TNC bar", Page 155 In the **Editor** operating mode and the **MDI** application, program a straight line **L** with the actual positions of all axes
Q FN	In the **Editor** operating mode, open the **FN** folder in the **Insert NC function** window
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
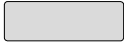
Key	Meaning
↑ ... → ▲ ... ▶	Position the cursor
GOTO	■ Position the cursor by using the block number of an NC block ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one 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

Poten-tiometer	Function
100 50 150 0 F %	Increase or reduce the feed rate **Further information:** "Feed rate F", Page 284
100 50 150 0 S %	Increase or reduce the spindle speed **Further information:** "Spindle speed S", Page 283

2.8.3 Keyboard shortcuts for operating the control

With a keyboard unit or a USB keyboard, you can use keyboard shortcuts in your control. In the User's Manual, the labels of the keys are used when indicating keyboard shortcuts. Keys without a label are indicated as follows:

Key	Designation
⇧	SHIFT
	SPACE
↵	RETURN
⇄	TAB
↑	UP
↓	DOWN
→	RIGHT
←	LEFT

2.8.4 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	Meaning
←	**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 or close the **Calculator**
⌨	Open or close the **Screen keyboard**
⚙	Open or close the **Settings** selection menu
»	**Open or close** ▪ White: expand the TNC bar or machine manufacturer's bar ▪ Green: collapse the TNC bar or machine manufacturer's bar ▪ Gray: Confirm message
+	**Add**
▯	**Open**
✕	**Close**
▢	**Maximize**
⊡	**Reduce**
⋮	**Move** Change the position of workspaces or windows
⋰	**Scale** Resize windows
. . .	File functions are available

Icon or shortcut	Meaning
⭐	▪ Black: **Add favorite** ▪ Yellow: **Remove favorite**
💾 **CTRL + S**	**Save**
📝	**Save as**
🔍 **CTRL + F**	**Find**
✂️ **CTRL + X**	**Cut**
📄 **CTRL + C**	**Copy**
📋 **CTRL + V**	**Paste**
↩ **CTRL + Z**	**Undo**
↪ **CTRL + Y**	**Redo**
≡▼	Open or close the selection menu ⓘ The control groups the icons of the title bar depending on the size of the workspace in a selection menu.
≡▲	
▦	Open or close the **Workspaces** selection menu
⚠	Show the **Message menu**

2.8.5 The Desktop menu **workspace**

Application

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

Description of function

The title bar of the **Desktop menu** workspace includes the following functions:

- The **Active Configuration** selection menu
  Using the selection menu, you can activate a configuration of the control interface.
  Further information: "Configuring the control's user interface", Page 1838
- Full-text search
  Search for functions in the workspace with the full-text search.
  Further information: "Adding and removing favorites", Page 110

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

- **Control**
  In this area you can open operating modes or applications.
  Further information: "Overview of the operating modes", Page 93
  Further information: "Overview of workspaces", Page 96
- **Tools**
  In this area you can open some tools from the HEROS operating system.
  Further information: "HEROS Operating System", Page 1867
- **Help**
  In this area you can open training videos or **TNCguide**.
  Further information: "User's Manual as integrated product aid: TNCguide", Page 66
- **Favorites**
  In this area you will find the favorites that you have chosen.
  Further information: "Adding and removing favorites", Page 110
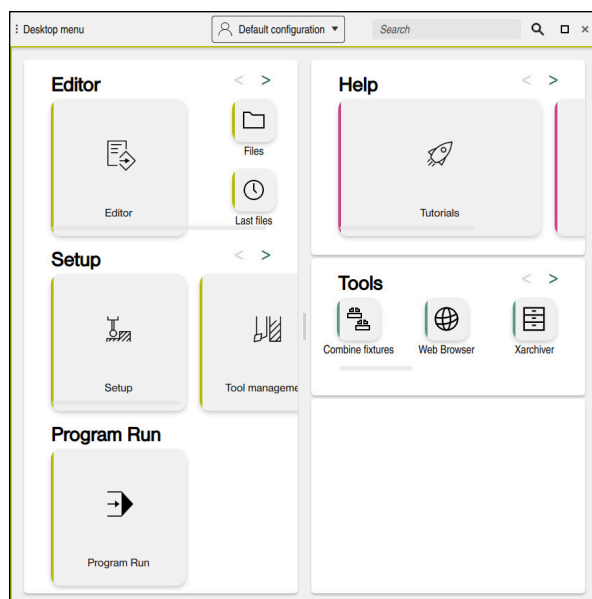
The **Desktop menu** workspace

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

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 on the machine and the control
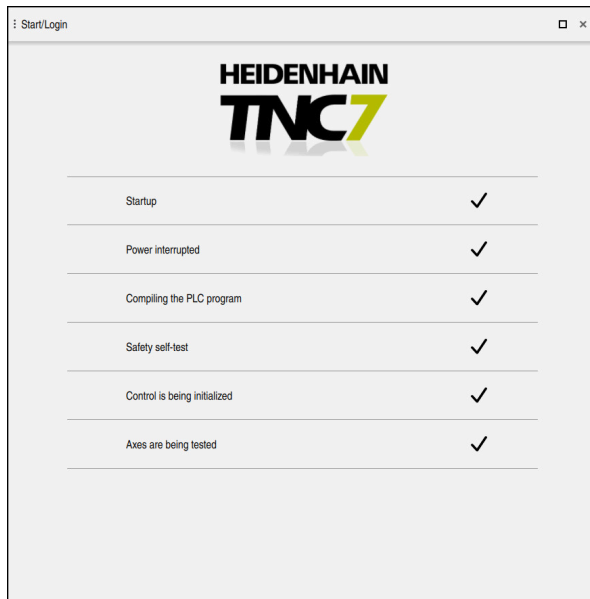
The **Start/Login** workspace

⚠️ **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 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 shows 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:** "The Referencing workspace", Page 181

▶ 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:** "The Manual operation application", Page 186

More detailed information

■ Switching on and off

   Further information: "Powering On and Off", Page 177

■ Position encoders

   Further information: "Position encoders and reference marks", Page 195

■ Axis reference run

   Further information: "The Referencing workspace", Page 181

3.3 Programming and simulating a workpiece

3.3.1 Example task 1338459
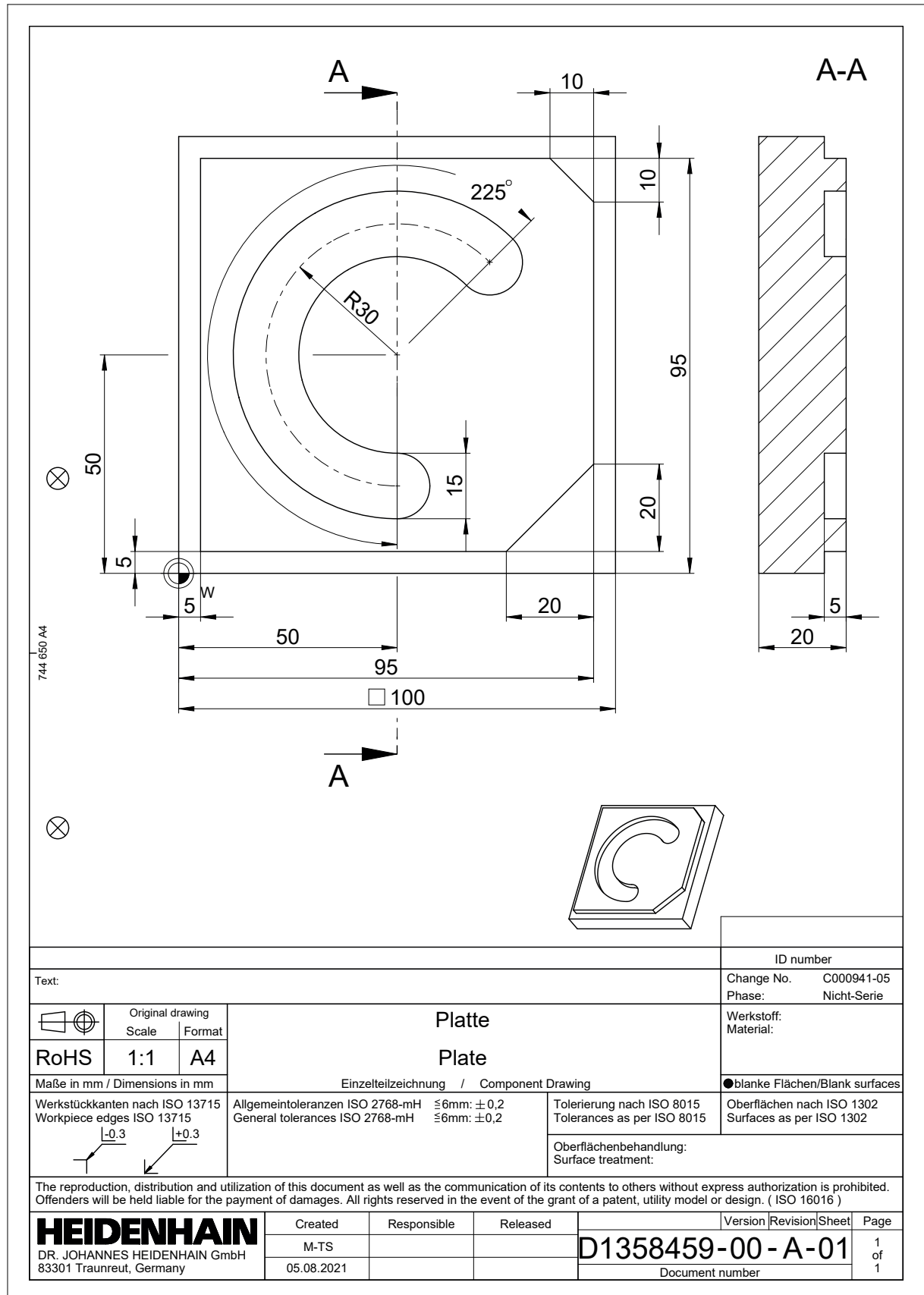
3.3.2   Selecting the Editor operating mode

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

■ The **Editor** operating mode

Further information: "The Editor operating mode", Page 202

3.3.3   Configuring the control's user interface for programming

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

ℹ️   The first steps describe the procedure when you are in the **Klartext editor** mode with the **Form** column 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: "Inserting and editing NC functions", Page 217

■ The **Form** column

Further information: "The Form column in the Program workspace", Page 214

3.3.4 Creating a new NC program
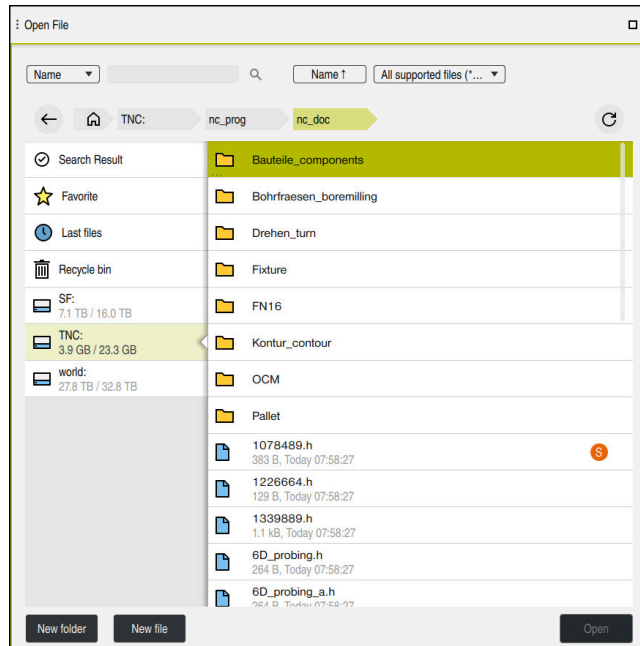
The **Open File** workspace in the **Editor** operating mode

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

▶ Select **New file**

▶ Enter a file name (e.g., 1338459.h)
▶ Confirm with the **ENT** key

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

More detailed information

■ The **Open File** workspace
Further information: "The Open File workspace", Page 834
■ The **Editor** operating mode
Further information: "The Editor operating mode", Page 202

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 you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

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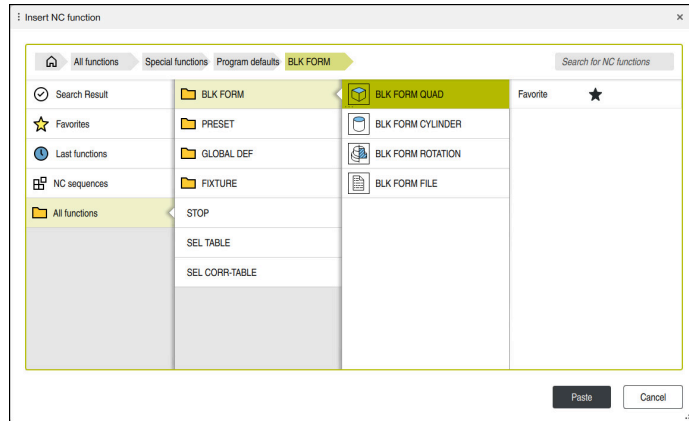
The **Insert NC function** window for workpiece blank definition

Defining a cuboid workpiece blank
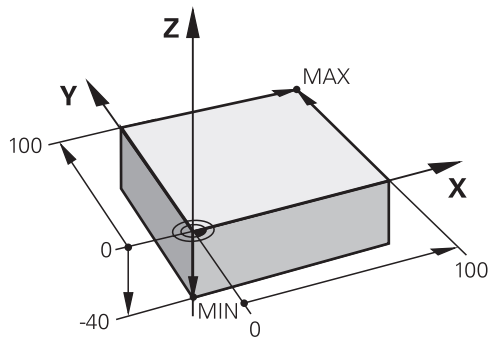
Cuboid workpiece blank with minimum point and maximum point

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

> ℹ️ 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**
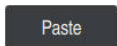
▶ Select **Paste**
> The control inserts the NC block for definition of the workpiece blank.

▶ 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

▶ Select **Confirm**
> The control concludes the NC block.
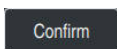
The **Form** column with the defined columns

| 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 |

⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

More detailed information

- Inserting the workpiece blank
  Further information: "Defining a workpiece blank with BLK FORM", Page 244
- Reference points in the machine
  Further information: "Presets in the machine", Page 196

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

> ℹ 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:
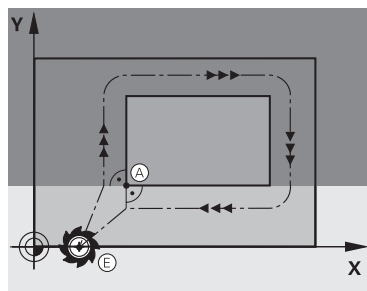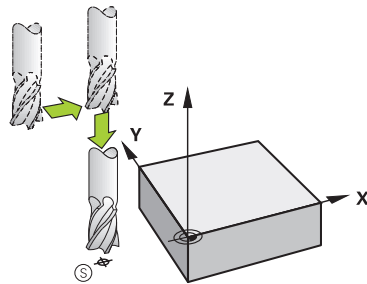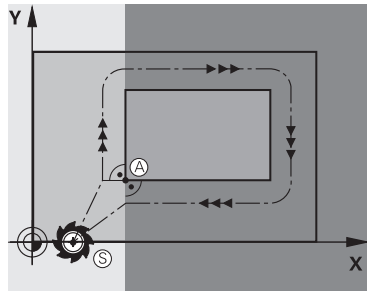
Help graphic	Position
	Starting point The following preconditions apply for the starting point: ■ No tool radius compensation ■ Approachable without danger of collision ■ Near to the first contour point The graphic shows the following information: If you define the starting point to be in the dark gray 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.
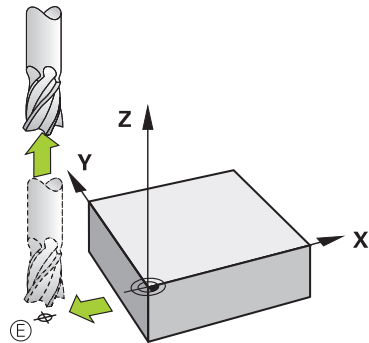
Help graphic	Position
	Departing from the end point in the tool axis Program the tool axis separately when departing from the end point.
	Identical starting and end points Do not program any tool radius compensation if the starting point and end point are the same. In order to make sure that the contour 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: "Fundamentals of approach and departure functions", Page 330

3.3.8 Programming a simple contour
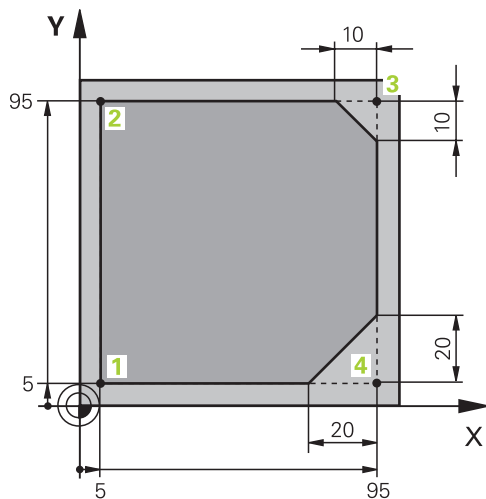
Workpiece to be programmed

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 118

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.

> **i** 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

The **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 12 Z S6500
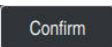
⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

Move the tool to a safe position
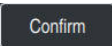
The **Form** column with the syntax elements of a straight line

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

▶ 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)
▶ Select **Confirm**
> The control concludes the NC block.
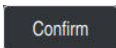
6 L Z-5 R0 F3000 M8

Approaching the contour
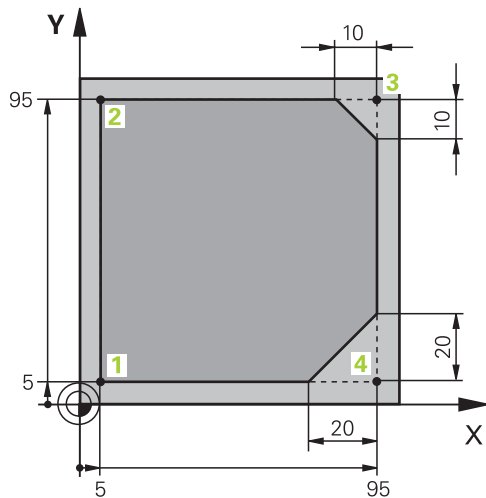
Workpiece to be programmed

The **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 the 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
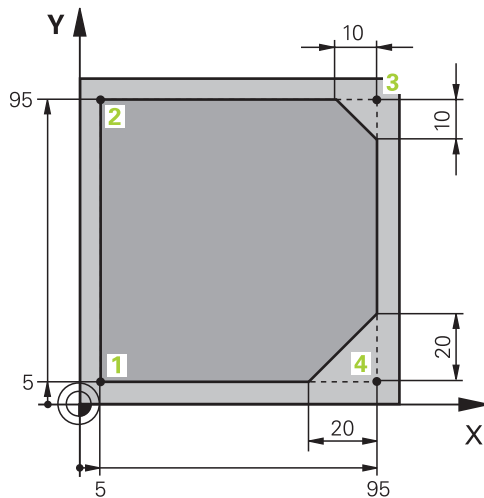
Workpiece to be programmed

To machine the contour:

▶ Select the path function **L**
▶ Enter the coordinates of contour point **2** that differ (e.g., **Y 95**)
▶ Conclude the NC block with **Confirm**
> The control applies the changed value and retains all of the other information from the previous NC block.
▶ Select the path function **L**
▶ Enter the coordinates of contour point **3** that differ (e.g., **X 95**)
▶ Conclude the NC block with **Confirm**
▶ Select the path function **CHF**
▶ Enter the chamfer width (e.g., **10**)
▶ Conclude the NC block with **Confirm**
▶ Select the path function **L**
▶ Enter the coordinates of contour point **4** that differ (e.g., **Y 5**)
▶ Conclude the NC block with **Confirm**
▶ Select the path function **CHF**
▶ Enter the chamfer width (e.g., **20**)
▶ Conclude the NC block with **Confirm**
▶ Select the path function **L**
▶ Enter the coordinates of contour point **1** that differ (e.g., **X 5**)
▶ 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

The **Form** column with the syntax elements of a departure function

To depart from the contour:
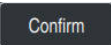
APPR
/DEP
> ► Select the **APPR DEP** path function
> ＞ The control opens the **Insert NC function** window.

> ► 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 279
■ Line **L**
Further information: "Straight line L", Page 300
■ Designation of the axes and the working plane
Further information: "Designation of the axes of milling machines", Page 194
■ Functions for approaching and departing from the contour
Further information: "Fundamentals of approach and departure functions", Page 330
■ Chamfer **CHF**
Further information: "Chamfer CHF", Page 302
■ Miscellaneous functions
Further information: "Overview of miscellaneous functions", Page 965

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 115

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:

TOOL CALL	▶ 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**)
Confirm	▶ Select **Confirm**
	> The control concludes the NC block.

16 TOOL CALL 6 Z S6500

Moving the tool to a safe position

The **Form** column with the syntax elements of a straight line

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**
	> 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**

▶ Select **X**

▶ Enter a value (e.g., **+50**)

▶ Select **Y**

▶ Enter a value (e.g., **+50**)

▶ Select the **FMAX** feed rate

▶ Select **Confirm**

> The control concludes the NC block.

18 L X+50 Y+50 FMAX
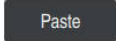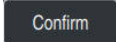
Defining a cycle

The **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.

▶ Select Cycle **254 CIRCULAR SLOT**

Paste

▶ Select **Paste**
> The control inserts the cycle.
▶ Open the **Form** column
▶ 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:

▶ 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**, 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

■ Working with cycles
Further information: "Working with cycles", Page 222

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

■ The **Simulation** workspace
Further information: "The Simulation Workspace", Page 1197

3.3.11 Simulating an NC program

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

Starting the simulation
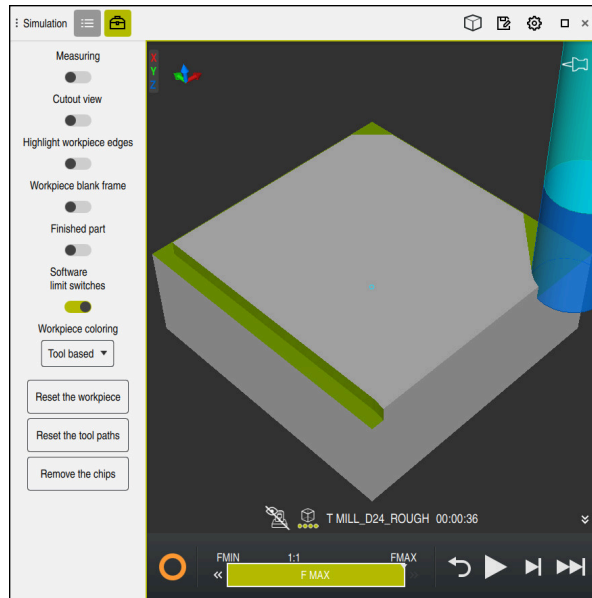
The **Simulation** workspace in the **Editor** operating mode

To start the simulation:

▶ ▶ Select **Start**
> The control might ask whether the file should be saved.

Save ▶ 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

- The **Simulation** workspace
  Further information: "The Simulation Workspace", Page 1197

3.4 Configuring a tool

3.4.1 Selecting the Tables operating mode

You can 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

- The **Tables** operating mode
  Further information: "The Tables operating mode", Page 1666

3.4.2 Configuring the control's user interface

The **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

- The **Form** workspace
  Further information: "The Form workspace for tables", Page 1676
- The **Table** workspace
  Further information: "The Table workspace", Page 1670

3.4.3 Preparing and measuring tools

To prepare tools:
- ▶ Clamp the required tools in their tool holders
- ▶ Measure the tools

  Further information: "Measuring the tool by scratching", Page 1284

- ▶ Write down the length and the radius or transfer these directly to the control

3.4.4 Editing within tool management

The **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

- ▶ Enable **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

■ The **Tables** operating mode
Further information: "The Tables operating mode", Page 1666

■ The **Form** workspace
Further information: "The Form workspace for tables", Page 1676

■ Tool management
Further information: "Tool management ", Page 269

■ Tool types
Further information: "Tool types", Page 261

3.4.5 Editing the pocket table

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

The **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
  - ▶ Enable **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 1698

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 the operating modes", Page 93

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:

▶ 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.

▶ Select **Compensate the active preset**

> The control applies the calculated results to the workpiece preset.

> The control highlights the line with a preset symbol.

▶ Select **Exit probing**

> The control closes the probing cycle.
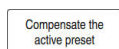
The **Probing function** workspace with an open manual probing function

More detailed information

- The **Probing function** workspace
  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257
- Reference points in the machine
  Further information: "Presets in the machine", Page 196
- Tool change in the **Manual operation** application
  Further information: "The Manual operation application", Page 186

3.6 Machining a workpiece

3.6.1 Selecting an operating mode

You can 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

■ The **Program Run** operating mode
Further information: "The Program Run operating mode", Page 1638

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
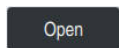
Open

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

More detailed information

■ The **Open File** workspace
Further information: "The Open File workspace", Page 834

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 regardless of the control's status!

▶ Always shut down the control

▶ Only operate the main switch after being prompted on the screen

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**

> If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.

▶ If necessary, save unsaved NC programs with **Save** or **Save as**

> The control shuts down.

> After completion of the shutdown process, the control displays the text **Now you can switch off.**

▶ Switch off the main power switch of the machine

4

Status Displays

4.1 Overview

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: "The Positions workspace", Page 149

- Status overview on the TNC bar

  Further information: "Status overview on the TNC bar", Page 155

- Additional status displays for specific areas in the **Status** workspace

  Further information: "The Status workspace", Page 157

- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece

  Further information: "The Simulation status workspace", Page 170

4.2 The 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
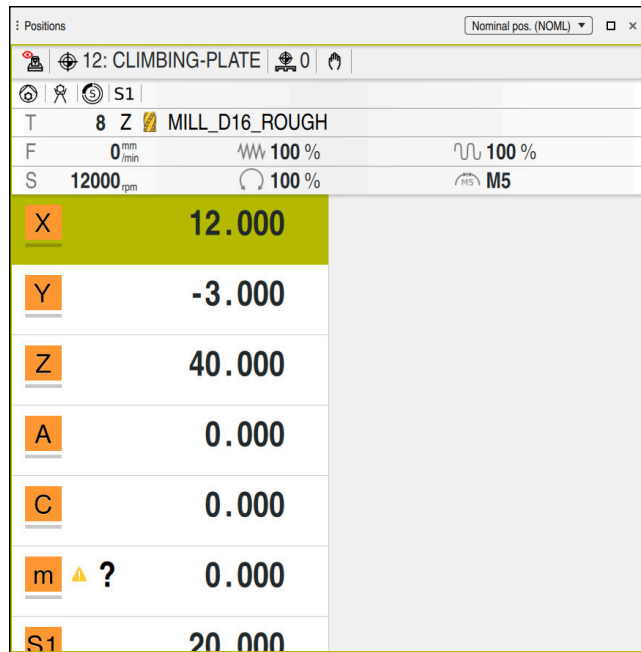
The **Positions** workspace with general status display

You can open the **Positions** workspace in the following operating modes:
- **Manual**
- **Program Run**

Further information: "Overview of the operating modes", Page 93

The **Positions** workspace provides the following information:
- Icons of active and inactive functions (e.g., Dynamic Collision Monitoring DCM (#40 / #5-03-1))
- 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 1774
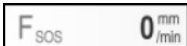
Axis display and position display

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

Icon	Meaning
IST	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 172
X	Axes The X axis is selected. You can move the selected axis.
m	The auxiliary axis **m** is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. **Further information:** "Definition", Page 154
?	The axis is not referenced.
⚠	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 1775
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.
⊛	You cannot move the axis with the handwheel. > ⚙ Refer to your machine manual. > The machine manufacturer defines which axes you can move with the handwheel.
F SOS 0 mm/min	Feed status when stopped **Further information:** "Functional safety FS in the Positions workspace", Page 1772
S SOS 0 U/min	Spindle status when stopped **Further information:** "Functional safety FS in the Positions workspace", Page 1772

Presets and technology values

Icon	Meaning
⊕	Number and comment of the active workpiece preset The number corresponds to the active row number of the preset table. The comment corresponds to the content of the **DOC** column. **Further information:** "Preset management", Page 705
⊕ (pallet)	Number of the active pallet preset The number corresponds to the active row number in the pallet preset table. **Further information:** "Pallet preset table", Page 1635
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. ■ If **M136** is active: active feed rate in mm/rev **Further information:** "Interpreting the feed rate as mm/rev with M136", Page 991 ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 105 If a feed-rate limitation has been activated with the **F LIMIT** button, the area is labeled **F LIMIT** instead of **F**. The control displays the text **F LIMIT** and the feed-rate value in orange. **Further information:** "Feed rate limit F LIMIT", Page 1642
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:** "The Program Run operating mode", Page 1638
	RL tool radius compensation is active. **Further information:** "Tool radius compensation", Page 798
	RR tool radius compensation is active. **Further information:** "Tool radius compensation", Page 798 These symbols are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1649
	R+ tool radius compensation is active. **Further information:** "Tool radius compensation", Page 798
	R- tool radius compensation is active. **Further information:** "Tool radius compensation", Page 798 These symbols are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1649
	3D tool compensation is active (#9 / #4-01-1). **Further information:** "3D tool compensation (#9 / #4-01-1)", Page 805 This symbol is transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1649
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 707
	The basic rotation will be taken into account while moving the axes. **Further information:** "Selection item Basic rotation", Page 783
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 707
	The tilted working plane will be taken into account while moving the axes. **Further information:** "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737

Icon	Meaning
	Further information: "The 3D ROT selection item", Page 784
	The **Tool axis** function is active (#21 / #4-02-1). **Further information:** "The Tool axis selection item", Page 783
	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 717 **Further information:** "Mirroring with TRANS MIRROR", Page 729
S %	The pulsing spindle speed function **S-PULSE** is active. **Further information:** "Pulsing spindle speed with FUNCTION S-PULSE", Page 895
	The **PARAXCOMP DISPLAY** function is active.
	The **PARAXCOMP MOVE** function is active. **Further information:** "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 933
	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 938
TCPM	The function **M128** or **FUNCTION TCPM** is active (#9 / #4-01-1). **Further information:** "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
	The Dynamic Collision Monitoring function (DCM) is active (#40 / #5-03-1).
	The Dynamic Collision Monitoring function (DCM) is not active (#40 / #5-03-1). **Further information:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
	The Dynamic Collision Monitoring function (DCM) is active with a reduced minimum distance (#140 / #5-03-2). **Further information:** "Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)", Page 878
AFC L	The Adaptive Feed Control function (AFC) is active in teach-in cut mode (#45 / #2-31-1).
AFC	The Adaptive Feed Control function (AFC) is active in closed-loop mode (#45 / #2-31-1). **Further information:** "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

Icon	Meaning
ACC	The Active Chatter Control function (ACC) is active (#145 / #2-30-1). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 893

> **i** 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) (#40 / #5-03-1) 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 TNC bar

Application

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

Description of function

General information
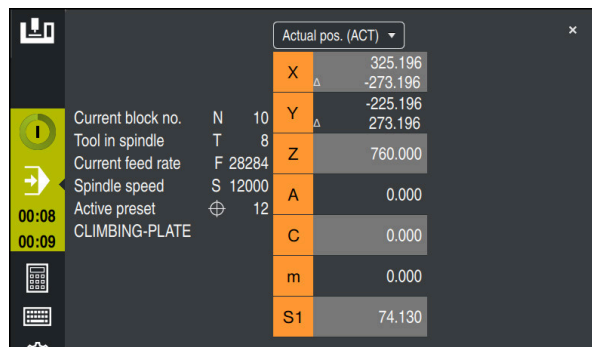
Status overview of the TNC bar with open position display

While an NC program or individual NC blocks are being executed, the control displays the following information in the status overview:

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

  Further information: "Definition", Page 156

- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time

  The control displays the run times of the NC program in mm:ss format. As soon as an NC program run time exceeds 59:59, the control changes the format to hh:mm.

  > ⓘ The control displays the same value for the program run time as on the **PGM** tab of the **Status** workspace.
  >
  > In the **Status** workspace the control shows the program run time in hh:mm:ss format.
  >
  > **Further information:** "Display of the program run time", Page 171

- Active tool
- Active feed rate
- Current spindle speed
- Number and comment of the active workpiece preset
- Position display

Position display

If you select the status overview area, the control opens or closes the position display with the current axis positions. The position display mode can be selected independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).

Further information: "The Positions workspace", Page 149

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. During programming, you can thus transfer the values directly into a programming dialog.

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 1643

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

4.4 The 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 the operating modes", Page 93

Icons

The following icons are shown in the **Status** workspace:

Icon	Meaning
⬚	**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
⚙	**Settings** Some areas have their own settings. Use this icon to customize the contents of the area (e.g., by defining the variable range to be displayed).
⭐	**Favorite** **Further information:** "The Favorites tab", Page 158

➕	**Add**
	 The control only shows this icon when you are adapting the layout.
	 With this icon you can add the following elements:
	 ■ Column
	    You can divide the workspace into several columns.
	    Further information: "Adding a column in the workspace", Page 1678
	 ■ Area
	    In the **Favorites** view you can add another area.

➖	**Remove** The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

The Favorites tab

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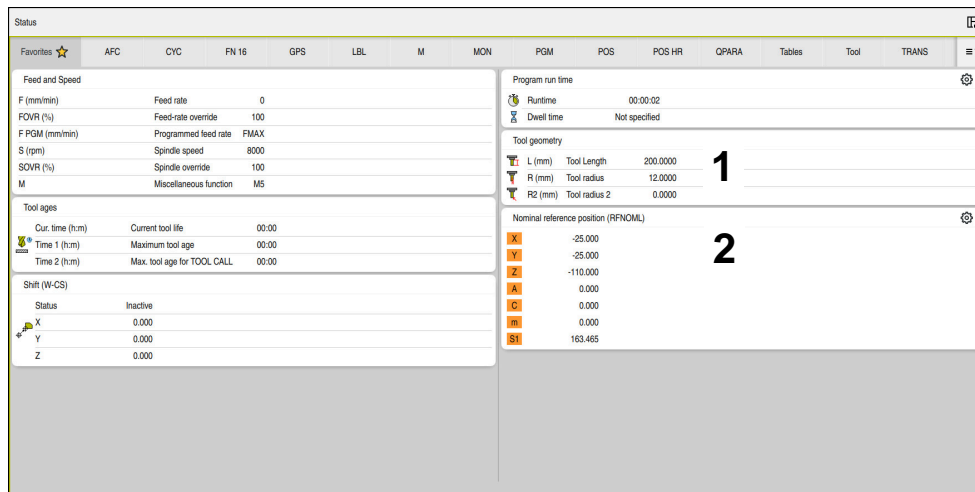
The **Favorites** tab

1　Area
2　Contents

Each area of the status display has its own **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

The AFC tab (#45 / #2-31-1)

The control displays information on the Adaptive Feed Control function (AFC) (#45 / #2-31-1) on the **AFC** tab.

Further information: "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

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 (#45 / #2-31-1)", Page 887 ■ **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 **CYCL DEF** function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLER-ANCE	■ **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** limited by Dynamic Collision Monitoring (DCM) **TOLERANCE** (#40 / #5-03-1)

⚙️ The machine manufacturer defines the tolerance limits using Dynamic Collision Monitoring (DCM) (#40 / #5-03-1).

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.

The FN 16 tab

On the **FN 16** tab, the control displays the contents of a file output to the screen with **FN 16: F-PRINT**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1028

Area	Contents
Output	Contents of an output file that was output with **FN 16: F-PRINT**, such as measured values or texts.
	To stop the output:
	■ Defining the **SCLR:** output path (Screen Clear)
	■ Select the **Clear** button
	■ Select the **Reset program** button
	■ Select a new NC program

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 360

Area	Contents
Subprogram calls	▪ **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 963

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.

The MON tab (#155 / #5-02-1)

On the **MON** tab, the control displays information on monitoring of defined machine components using the Component monitoring function (#155 / #5-02-1).

Further information: "Component monitoring with MONITORING HEATMAP (#155 / #5-02-1)", Page 908

> ⚙ Refer to your machine manual.
>
> The machine manufacturer specifies which machine components are monitored, and to what extent.
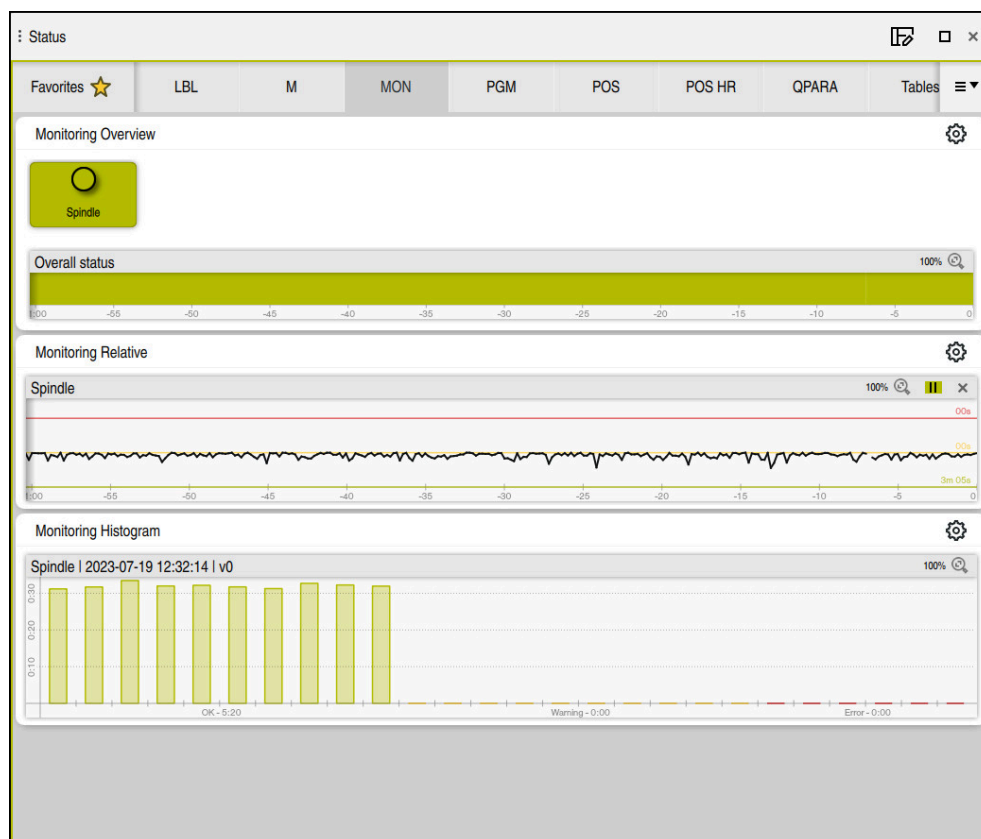
The **MON** tab with configured spindle speed monitoring

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.
	If a component cannot be monitored, the control displays a gray icon. A component cannot be monitored, for example, if its configuration is missing or is wrong.
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 will be shown by the control.

Area	Contents
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 the graphical representation 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 1058
Program run time	■ **Runtime** Run time of the NC program in hh:mm:ss format ■ **Dwell time** Countdown of the waiting time in seconds from the following functions: ■ **FUNCTION DWELL** ■ 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 171
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**
Radius compensation	Programmed tool radius compensation
Program run options	Active breakpoints in connection with the override controller **Further information:** "Override Controller", Page 1756

POS tab

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

Area	Contents
Position display, for example **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 172
Feed and Speed	■ Active **Feed** in mm/min If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the **F LIMIT** button, the control displays **LIMIT** in square brackets. **Further information:** "Feed rate limit F LIMIT", Page 1642 If the feed rate is limited using the **F limited** button, the control displays the active safety function in square brackets. **Further information:** "Safety functions", Page 1771 ■ Active **Feed-rate override** in % ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min If **M136** is active: active feed rate in mm/rev **Further information:** "Interpreting the feed rate as mm/rev with M136", Page 991 ■ 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 (#8 / #1-01-1)", Page 737 If axis angles are active, the control displays in this area only the values of the physically present axes. Defined values in the **3-D rotation** window **Further information:** "The 3D ROT selection item", Page 784
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 705
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1

Area	Contents
	Traverse ranges are machine-specific. If no traverse range is active, then **Traverse range not defined** is displayed in this area.
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)** If you use **M118** (#21 / #4-02-1), handwheel superimpositioning is always effective in the machine coordinate system **M-CS**. **Further information:** "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979
Handwheel superimp.	■ **Max. val.** Maximum value of the individual axes, programmed in **M118** (#21 / #4-02-1) ■ **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 1006

You can use the **Parameter list** window to define which variables the control shows in the individual areas. Up to 22 variables can be displayed in each area.

Further information: "Defining the contents of the QPARA tab", Page 175

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

The 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 ■ Preset table ■ Datum table ■ Pocket table ■ Touch-probe table (#17 / #1-05-1)

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 ■ Row 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 727
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** "Mirroring with TRANS MIRROR", Page 729 **Further information:** "Cycle 8 MIRRORING", Page 717
Active angle of rotation	Rotation angle defined with either the **TRANS ROTATION** function or Cycle **10 ROTATION** **Further information:** "Rotations with TRANS ROTATION", Page 732 **Further information:** "Cycle 10 ROTATION ", Page 718
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737
Center of scaling	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 721
Active scaling factors	Scaling factors that were defined for the individual linear axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Scaling with TRANS SCALE", Page 733 **Further information:** "Cycle 11 SCALING FACTOR ", Page 720 **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 721
Shift (WPL-CS)	Active shift in the working plane coordinate system **WPL-CS** by means of the **FUNCTION CORRDATA** function **Further information:** "Activating a compensation value with FUNCTION CORRDATA", Page 804
Table	■ Path of the selected compensation table ***.wco** ■ Row number of the selected compensation table ***.wco** ■ Content of the **DOC** column of the active row **Further information:** "Compensation table *.wco", Page 1734

TT tab

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

Further information: "Hardware enhancements", Page 89

Area	Contents
TT: tool measurement	■ **T** Tool number ■ **Name** Tool name ■ **Measuring method** Selected measurement method for tool measurement (e.g., **Length**) ■ **Min (mm)** When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge. **Further information:** "Definitions", Page 169 ■ **Max (mm)** When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge. ■ **DYN Rotation (mm)** When measuring milling cutters with a rotating spindle, the control shows values in this area. ⓘ 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.
TT: measurement of individual teeth	**Number** List of the measurements performed and the measured values of the individual cutting edges

The Tool tab

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

Further information: "Tool types", Page 261

Contents for milling tools

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 With **Program**, the control displays the values from a tool call with **TOOL CALL** or from a tool compensation with a compensation table ***.tcs**. **Further information:** "Tool call", Page 279 **Further information:** "Tool compensation with compensation tables", Page 801 With **Table**, the control displays the values from the tool management. **Further information:** "Tool management ", Page 269
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**)

Area	Contents
	■ **Type** Tool type of the active tool (e.g., **DRILL**)

Definitions

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 The 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: "The Favorites tab", Page 158
- **CYC**
  Further information: "CYC tab", Page 160
- **FN 16**
  Further information: "The FN 16 tab", Page 160
- **LBL**
  Further information: "LBL tab", Page 161
- **M**
  Further information: "M tab", Page 161
- **PGM**
  Further information: "PGM tab", Page 163
- **POS**
  Further information: "POS tab", Page 164
- **QPARA**
  Further information: "QPARA tab", Page 165
- **Tables**
  Further information: "The Tables tab", Page 165
- **TRANS**
  Further information: "TRANS tab", Page 166
- **TT**
  Further information: "TT tab", Page 167
- **Tool**
  Further information: "The Tool tab", Page 168

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**. The control takes traversing movements and dwell times into account.

In addition, the control calculates the remaining run time of the NC program.

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
- The **Simulation** workspace in the **Editor** operating mode

Use the **Settings** button in the **Program run time** area to influence the calculated program run time.

Further information: "PGM tab", Page 163

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

Remaining run time of the NC program

If a tool usage file is available, the control calculates for the **Program Run** operating mode the duration of executing the active NC program. During program run, the control updates the remaining run time.

Further information: "Tool usage test", Page 287

The control shows the remaining run time in the status overview on the TNC bar.

The control does not take the feed-rate potentiometer setting into account, but calculates with a feed rate of 100%.

The following functions reset the remaining run time:

- Selecting a new NC program for program run
- **Internal stop** button
- Generate new tool usage file

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

Further information: "Interrupting, stopping or canceling program run", Page 1643

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:

- The **Positions** workspace
- Status overview on the control bar
- The **POS** tab of the **Status** workspace
- The **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.
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 (#21 / #4-02-1)", Page 979

> **⚙** 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 (e.g., **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 150

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 will show.

Further information: "QPARA tab", Page 165

To define the contents of the **QPARA** tab:
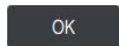
> ▶ Select the **QPARA** tab

⚙ ▶ 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.

ℹ
- 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 195

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 opens 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: "The Referencing workspace", Page 181

Further information: "The Manual operation application", Page 186

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 shows the **Power interrupted** dialog in the **Start/Login** workspace.
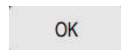
OK	▶ Press **OK** > The control compiles the PLC program.
Ⓘ	▶ Switch the machine control voltage on > The control checks the functioning of the emergency stop circuit. > If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation. > If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application. **Further information:** "The Referencing workspace", Page 181
⟳	▶ 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:** "The Manual operation application", Page 186

ⓘ   If startup is delayed by functional safety, the control displays the text **Functional safety requires input**. When you select the **FS** button, the control switches to the **Functional safety** application.

Further information: "The Functional safety application", Page 1772

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 manufacturer

5.2 The 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.
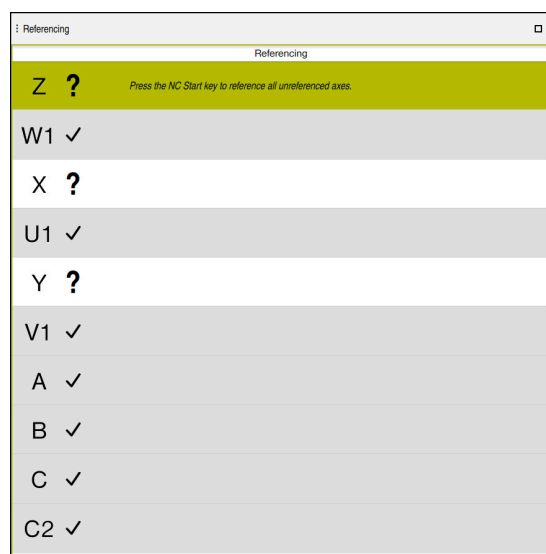
The **Referencing** workspace with axes to be referenced

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** (#8 / #1-01-1) function was active before the control was shut down, then the control will automatically reactivate this function after the restart. 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 working plane** function; otherwise, the control will interrupt the process with a warning. You can also home axes that are not activated in the current kinematic model without needing to deactivate **Tilt working plane**, such as a tool magazine.

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

5.3 Powering off

Application

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

Description of function

You can 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.

If NC programs or contours contain any unsaved changes, the control displays the unsaved changes in the **Close file** window. You can save the changes, discard them, or cancel the shutdown.

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**
> If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
▶ If necessary, save unsaved NC programs with **Save** or **Save as**
> The control shuts down.
> After completion of the shutdown process, the control displays the text **Now you can switch off.**
▶ Switch off the main power 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 regardless of the control's status!
▶ 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 that are active on the control might delay the shutdown, such as a connection to **Remote Desktop Manager** (#133 / #3-01-1)
   Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

6

Manual Operation

6.1 The 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 188

■ Incremental jog positioning of machine axes
Further information: "Incremental jog positioning of axes", Page 190

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 toggle 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 1741
M	Define a miscellaneous function **M** or use the selection menu to choose one and activate it with the **NC start** key. **Further information:** "Miscellaneous Functions", Page 963 The machine manufacturer uses the optional machine parameter **forbidManual** (no. 103917) to define which miscellaneous functions are allowed in the **Manual operation** application and are available in the selection menu.
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 283
F	Define the feed rate **F** and activate it with the **OK** button. **Further information:** "Feed rate F", Page 284
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 279
3D ROT	The control opens a window for the 3D rotation settings (#8 / #1-01-1). **Further information:** "The 3-D rotation window (#8 / #1-01-1)", Page 781
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:** "The Q parameter list window", Page 1010
DCM	The control opens the **Dyna. Coll. Monitoring (DCM)** window where you can activate or deactivate Dynamic Collision Monitoring (DCM (#40 / #5-03-1)). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 853
Manual cycles	The machine manufacturer can define manual cycles that you can use by means of this button.
F limited	Use this option to activate or deactivate the feed-rate limit for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 1774
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 190
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 705

Button	Meaning
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 269
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:** "Tool management ", Page 269

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 (#17 / #1-05-1)", Page 1257

Related topics

■ Programming traverse movements

Further information: "Path Functions", Page 291

■ Executing traverse movements in the **MDI** application

Further information: "The MDI Application ", Page 1221

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 1741

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

Further information: "Status Displays", Page 147

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 TNC bar", Page 155

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

Incremental jog positioning allows you to move a machine axis by a preset distance.
The input range for the infeed is from 0.001 mm to 10 mm.
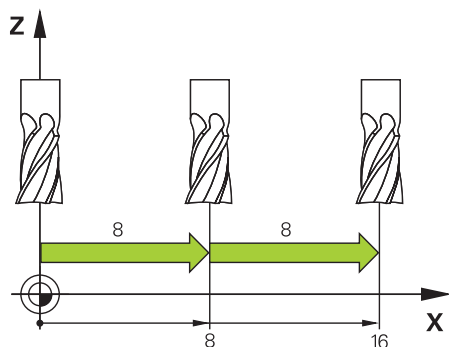
To position an axis incrementally:

▶ 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.

> **i**   You can also end incremental jog positioning with the **Off** button in the **Jog increment** area.
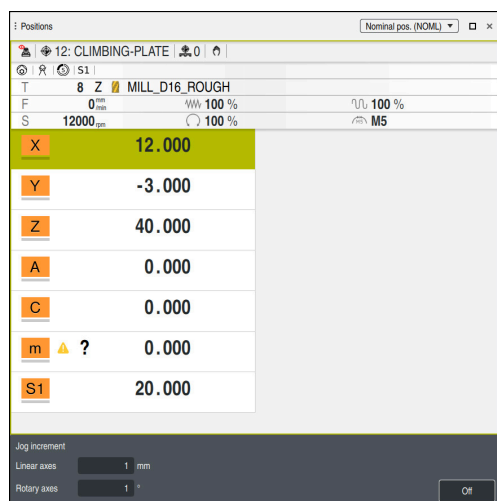
The **Positions** workspace with active **Jog increment** area

Note

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

7

NC and Programming Fundamentals

7.1 NC fundamentals

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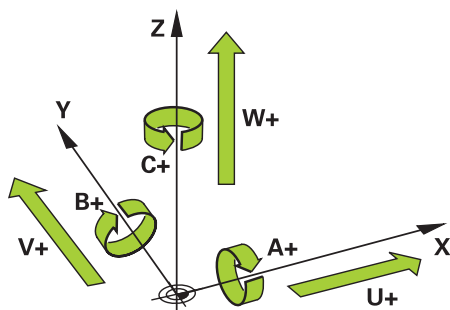
The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

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.

> The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.
> If the axis position does not change, you can nevertheless program more than four axes.

7.1.2 Designation of the axes of 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**

> The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

7.1.3 Position encoders and reference marks

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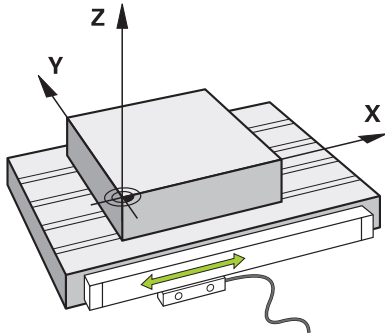
The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders. Rotary tables and rotary axes feature angle encoders.

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 690

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 181
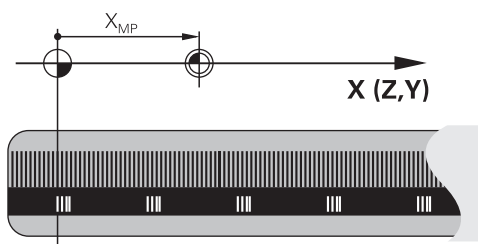
7.1.4 Presets in the machine

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 253

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 692 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 967
╬ 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 968
	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 195 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 181
⊕	**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 697 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 705 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 (#8 / #1-01-1), 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 296
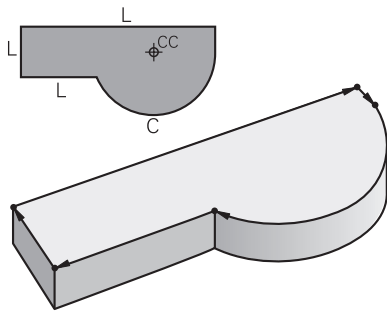
7.2.2 Graphical programming

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 1089

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 963

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 360

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: "The Q parameter list window", Page 1010

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 1006

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 915

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 117

■ NC programs using CAD files
 Further information: "CAM-generated NC programs", Page 947

■ Structure of an NC program for contour machining
 Further information: "Structure of an NC program", Page 120

Description of function

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

Further information: "The Program workspace", Page 203

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	; NC function for workpiece blank definition, consisting of two NC blocks
2 BLK FORM 0.2 X+50 Y+50 Z+0	
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 compo-nent	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: "Areas of the Insert NC function window", Page 216
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.

Syntax compo-nent	Meaning
	Further information: "Appearance of the NC program", Page 205
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

- NC functions can also consist of more than one NC block, such as **BLK FORM**.
- Using the machine parameter **linebreak** (no. 105404), you can define how the control will display multi-line NC functions.
- 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 199

7.3.2 The 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** operating mode presents the following workspaces if an NC program is open:
- **Help**
  Further information: "The Help workspace", Page 1158
- **Contour**
  Further information: "Graphical programming", Page 1089
- **Program**
  Further information: "The Program workspace", Page 203
- **Simulation**
  Further information: "The Simulation Workspace", Page 1197
- **Simulation status**
  Further information: "The Simulation status workspace", Page 170
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 1160

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

Further information: "The Job list workspace", Page 1620

Further information: "The Form workspace for pallets", Page 1628

If the software option Batch Process Manager (#154 / #2-05-1) is active, the entire functionality for executing pallet tables is available to you.

Further information: "The Job list workspace", Page 1620

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 1089
▤	The control uses this icon to show that a pallet table is open. **Further information:** "Pallet Machining and Job Lists", Page 1619
⮕	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution. When simulating the opened NC program, the control displays the execution cursor.
Klartext editor	If this toggle switch is active, then you are using dialog-guided programming. If this toggle switch is not active, then you are programming in the text editor. **Further information:** "Inserting and editing NC functions", Page 217
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Inserting and editing NC functions", Page 217
GOTO block number	The control selects the block number that you defined. **Further information:** "GOTO function", Page 1163
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:** "The Q parameter list window", Page 1010
/ Skip block Off/On	Hide NC blocks with **/**. NC blocks hidden with a **/** character will be ignored during program run as soon as the **Skip block** toggle switch is active. **Further information:** "Hiding NC blocks", Page 1165
; 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 1164
Edit	The control opens the context menu. **Further information:** "Context menu", Page 1174
Select in Program Run	The control opens the file in the **Program Run** operating mode. **Further information:** "Program Run", Page 1637
Start the simulation	The control opens the **Simulation** workspace and starts graphic simulation. **Further information:** "The Simulation Workspace", Page 1197

7.3.3 The 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
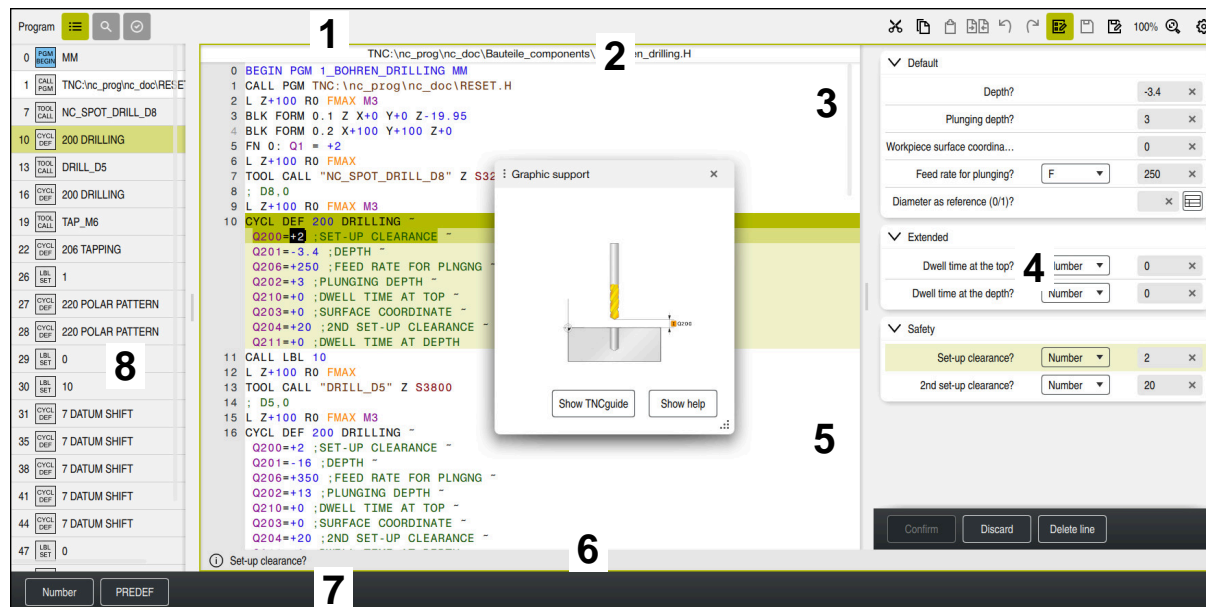
The **Program** workspace with active structure, help graphic, and form

1   Title bar

Further information: "Icons in the title bar", Page 205

2   File information bar

In the file information bar, the control shows the path and file name of the NC program. In the **Program Run** and **Editor** operating modes, the file information bar includes breadcrumb navigation.

Further information: "Navigation path in the Program workspace", Page 1646

3   Contents of the NC program

Further information: "Appearance of the NC program", Page 205

4   **Form** column

Further information: "The Form column in the Program workspace", Page 214

5   Help graphic of the syntax element being edited

Further information: "Help graphic", Page 206

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 possibilities for the syntax element being edited.

8   The **Structure**, **Search** or **Tool check** column

Further information: "The Structure column in the Program workspace", Page 1166

Further information: "The Search column in the Program workspace", Page 1169

Further information: "Tool usage test", Page 287

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 107

Icon or shortcut	Function
☰	Open and close the **Structure** column **Further information:** "The Structure column in the Program workspace", Page 1166
🔍 **CTRL + F**	Open and close the **Search** column **Further information:** "The Search column in the Program workspace", Page 1169
⊘	Open and close the **Tool check** column **Further information:** "Tool usage test", Page 287
▣▣	Activate and end comparison functions **Further information:** "Program comparison", Page 1172
▣✎	Open and close the **Form** column **Further information:** "The Form column in the Program workspace", Page 214
100%	Font size of the NC program > ℹ️ If you select the percent value, the control displays 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 206

Appearance of the NC program

By default the control shows the syntax with black characters. 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**
Gray	▪ Not to be executed **M1** miscellaneous function ▪ Not to be executed NC block hidden with a **/** character

Help graphic

When you are editing an NC block, the control shows for some NC functions a help graphic in a pop-up window that illustrates the current syntax element. If you change the size and position of the pop-up window, the control will save the settings separately for each tab.

Depending on the setting **Show help graphics automatically** or the machine parameter **stdTNChelp**, the control will display the help graphic as a pop-up window.

Further information: "Settings in the Program workspace", Page 206

The pop-up window includes the following buttons:

Button	Meaning
Show TNCguide	The control opens **TNCguide** at the corresponding position in the **Help** workspace. **Further information:** "User's Manual as integrated product aid: TNCguide", Page 66
Show help	The control opens the help graphic in the **Help** workspace. If the **Help** workspace is open, the control will always display the help graphic there.

Further information: "The Help workspace", Page 1158

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 or application. The **Program settings** window consists of the following areas:

Area	The Editor operating mode	The Program Run operating mode	MDI application
Structure	✓	✓	✓
Edit	✓	-	✓
Klartext	✓	-	✓
Tables	-	✓	-
FN 16	-	✓	-

The Structure area
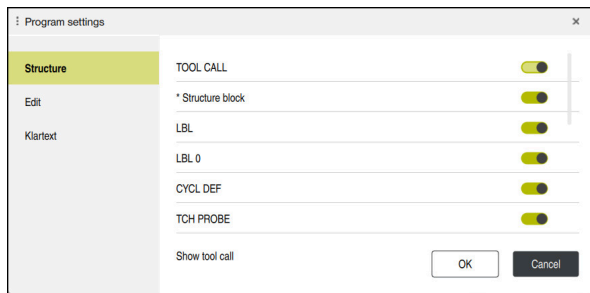
The **Structure** area in the **Program settings** window

In the **Structure** area, you can use toggle switches to choose which structuring items the control should display in the **Structure** column.

The following structure elements are available:

- **TOOL CALL**
- *** Structure block**
- **LBL**
- **LBL 0**
- **CYCL DEF**
- **TCH PROBE** (#17 / #1-05-1)
- **CALL PGM**
- **SEL PGM**
- **FUNCTION MODE**
- **M30 / M2**
- **M1**
- **M0 / STOP**
- **APPR / DEP**

The Edit area

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

Setting	Meaning
Automatic saving	**Save changes to the NC program automatically or manually**
	If the toggle 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 toggle switch is not active, you must save manually. Upon the stated actions, the control asks whether the changes should be saved.
Autocomplete in text mode	If the toggle switch is active, the control will automatically display a selection menu with possible syntax initiators or syntax elements when you select one of the following actions:
	■ Creating a new NC block
	■ Entering characters
	■ Pressing **CTRL + SPACE**
	If the toggle switch is not active, you can open the selection menu by pressing **CTRL + SPACE**.
	Further information: "Inserting NC functions", Page 218
Allow syntax errors in text mode	If the toggle switch is active, the control can save NC blocks in the text editor even if they contain syntax errors.
	If the toggle 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 219
Generate absolute paths	**Create relative or absolute path entries**
	If the toggle switch is active, the control uses absolute paths for called files, e.g.: **TNC:\nc_prog\\$mdi.h**.
	If the toggle 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 828
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 toggle 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 toggle switch is not active, the control does not format NC programs with more than 30 000 characters.
Back-up file when saving	If the toggle switch is active, the control will save a backup copy with the ***.h.bak** extension once you save the NC program.
	By removing the ***.bak** extension from the file name, you can restore the backup copy. The control overwrites the original file.

> ℹ When you select the **All Files (*.*)** filter, the control displays the file in the **Open File** workspace.

Setting	Meaning
	The same setting is also available in the machine parameter **createBackup** (no. 105401). The control will reconcile both setting options.
Behavior of the cursor after deletion of lines	If you activate the toggle switch and delete an NC program line, the cursor will move back to the previous NC block.
	The same setting is also available in the machine parameter **deleteBack** (no. 105402). The control will reconcile both setting options.
Show help graphics automatically	If the toggle switch is active, the control will show a help graphic in a pop-up window.
	The same setting is also available in the optional machine parameter **stdTNChelp** (no. 105405). The control will reconcile both setting options.
	When the **Help** workspace is open, the control will always display the help graphic there, independently of this setting.
	Further information: "The Help workspace", Page 1158
Confirmation request when deleting an NC block	If the toggle switch is active, the control will display a confirmation prompt in a pop-up window when you delete an NC block.
	The same setting is also available in the optional machine parameter **warningAtDEL** (no. 105407). The control will reconcile both setting options.
Comment blocks for NC sequences	If the toggle switch is active, the control adds a comment before and after each NC sequence.
	Each comment includes the following information:
	■ Start of the NC sequence
	■ Current date
	■ Current time
	■ Name of the NC sequence
	■ End of the NC sequence
	Further information: "NC sequences for reuse", Page 369
Hide NC functions that aren't available	If the toggle switch is active, the control will only display currently available NC functions in the **Insert NC function** window.
	If the toggle switch is not active, the control dims unavailable NC functions (e.g., for software options that are not enabled).
Put all path information in quotation marks	If the toggle switch is active, the control will automatically enclose path information in quotation marks when you select one of the following NC functions:
	■ **CALL PGM**
	■ Cycle **12 PGM CALL**
	■ **FN 16 F-PRINT**
	■ **FN 26 TABOPEN**
	The same setting is also available in the optional machine parameter **quotePaths** (no. 105414). The control will reconcile both setting options.
Display screen keyboard for editing	If a touchscreen is used, the control will display a context-sensitive virtual keyboard. A selection menu allows you to select the position of the virtual keyboard in the workspace or to hide the virtual keyboard.

Klartext area

In the **Klartext** area, select whether the control offers certain syntax elements of an NC block during input.

The control offers the following settings as toggle switches:

Setting	Meaning
Skip comment	If you activate this toggle switch, the control skips the comment function during programming for all NC functions. **Further information:** "Adding comments", Page 1164
Skip tool index	If you activate this toggle switch, the control skips the tool index for the following NC functions: ■ Call a tool with **TOOL CALL** **Further information:** "Tool call by TOOL CALL", Page 279 ■ Preselect a tool with **TOOL DEF** **Further information:** "Tool pre-selection by TOOL DEF", Page 286 **Further information:** "Indexed tool", Page 258
Skip linear superimposed interpolated axis values	If you activate this toggle switch, the control skips the **LIN_** syntax element for the following NC functions: ■ Circular contour **C** **Further information:** "Circular path C ", Page 308 ■ Circular contour **CR** **Further information:** "Circular path CR", Page 310 ■ Circular contour **CT** **Further information:** "Circular path CT", Page 313 **Further information:** "Linear superimpositioning of a circular path", Page 315

You can program the syntax elements in the form independently of the settings in the **Klartext** area.

Tables

In the **Tables** area, you can select a unique table for each of the application areas shown; this table is then active during program run.

Select the following tables using a selection window:

■ **Datums**
Further information: "Datum table *.d", Page 1721

■ **Tool correction**
Further information: "Compensation table *.tco", Page 1732

■ **Workpiece correction**
Further information: "Compensation table *.wco", Page 1734

FN 16

In the **FN 16** area, use the **Show pop-up window** toggle switch to select whether the control displays a window in conjunction with **FN 16**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1028

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 **ℹ** If you are working with a mouse, click with the right mouse key. **Further information:** "Context menu", Page 1174
✛	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 1177
⤢	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	Meaning
▲ ▼	■ 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 213
▶ ◀	■ Edit an NC block ■ During editing, navigate to previous or next syntax element
CTRL + RIGHT **CTRL + LEFT**	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 1163 ■ 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	Delete an NC block or cancel a dialog
END	■ Confirm entry and conclude an NC block ■ Open the **Add** tab
SHIFT + RETURN	Insert a line break in the Text editor mode Insert a line break in the **Form** column for comments
ESC	Cancel editing without applying changes
Klartext editor	Select **Klartext editor** mode or a text editor **Further information:** "Editing NC functions", Page 219
Insert NC function	Open the **Insert NC function** window **Further information:** "Areas of the Insert NC function window", Page 216
Edit	Open the context menu **Further information:** "Context menu", Page 1174

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.

> ℹ️ You can search for identical syntax initiators in an NC program. Select the syntax initiator by double-tapping or double-clicking it.

Notes

- When 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.
- If the NC block contains a syntax error, the control precedes the block number with a corresponding icon. Click the icon to see the associated error description.
- 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.
- If 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 adds this NC block.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is currently being executed in the **Program Run** operating mode.
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

The Form column in the Program workspace

Application

In the **Form** column of the **Program** workspace, the control shows all possible syntax elements for the currently selected NC function. In the form, you can edit all syntax elements as well as the syntax initiator, if required.

Related topics

■ The **Form** workspace for pallet tables

Further information: "The Form workspace for pallets", Page 1628

■ Editing an NC function in the **Form** column

Further information: "Editing NC functions", Page 219

Requirement

■ **Klartext editor** mode must be active

Description of function

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

Icon or button	Meaning
🖹	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 The **Insert NC function** window

Application

The **Insert NC function** window allows you to insert NC functions or NC sequences into an NC program.

Related topics

- Creating NC sequences
  Further information: "NC sequences for reuse", Page 369
- Inserting and editing NC functions
  Further information: "Inserting and editing NC functions", Page 217

Description of function

The **Insert NC function** window is available in the **Editor** operating mode and **MDI** application only.

> **i** In the **MDI** application, you can insert NC functions into the **$mdi.h** or **$mdi_inch.h** NC program only.

Areas of the Insert NC function window
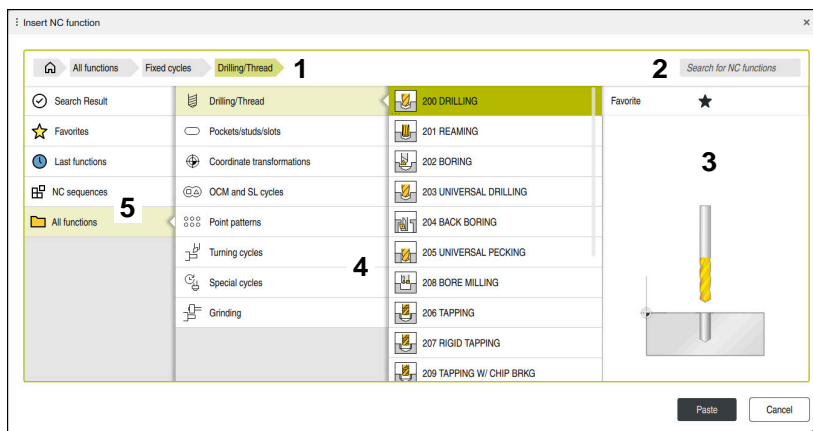
The **Insert NC function** window

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.

Further information: "Areas of file management", Page 825

2 Searching

Use the **Search for NC functions** feature to search for the syntax opener of the NC function or the name of the NC sequence.

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

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

3 The control shows the following information and functions:
- Add or remove a favorite
- Preview

    The control shows a preview of the content for NC sequences and a preview image for cycles.

4 Content columns

The control shows NC functions or folders that contain NC functions. The control displays up to two columns.

5 Navigation column

The navigation column contains the following areas:
- **Search Result**

    The control shows the following search results:
    - NC functions or miscellaneous functions with the content searched for in the name (e.g., Cycle **4019** when searching for "19")
    - Equivalent or alternative NC functions (e.g., **PATTERN DEF** when searching for "pattern")
    - Replacement functions for older and partly no longer offered functions (e.g., **PLANE** functions instead of Cycle **19**) **WORKING PLANE**
- **Favorites**

    The control displays all NC functions and NC sequences that you have marked as favorites.

    Further information: "Icons on the control's user interface", Page 107
- **Last functions**

The control shows the ten most recently used NC functions and NC sequences.

- **NC sequences**

  Use the NC sequences to insert a saved sequence of NC functions.

  Further information: "NC sequences for reuse", Page 369

- **All functions**

  The control shows all available NC functions in the folder structure.

  You can limit the selection possibilities using the keys or buttons. When you press the **CYCL DEF** key, the control will open the groups of cycles.

  Further information: "Keycaps for NC dialog", Page 102

In the **Search Result**, **Favorites** and **Last functions** areas, the control shows the path of the NC functions.

File functions in the Insert NC function window

If you drag an NC function to the right in the **Insert NC function** window, the control provides the following file functions:

- Add or remove a favorite
- Navigate to the NC function

  Not available in the **All functions** area

For NC sequences, the control provides the following additional file functions:

- Edit
- Rename
- Delete
- Activate or deactivate write protection
- Open the path in the **Files** operating mode

Further information: "NC sequences for reuse", Page 369

Notes

- 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.
- If software options are not enabled, the control dims unavailable contents in the **Insert NC function** window.

7.3.5 Inserting and editing NC functions

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 211

- **Insert NC function** window

  Further information: "The Insert NC function window", Page 215

Description of function

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

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

Inserting NC functions

The control provides the following options to insert 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 1160

■ Inserting an NC function by selecting it

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

Further information: "The Insert NC function window", Page 215

■ Inserting an NC function in the text editor

In the text editor, the control provides an auto-complete function when programming.

> 🛈 If text editor mode is active, the **Klartext editor** toggle switch is to the left and dimmed.

Further information: "Inserting NC functions", Page 218

Editing NC functions

The control provides the following options to edit NC functions:

■ Editing an NC function in the **Klartext editor** mode

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

■ Editing an NC function in the **Form** column

The **Form** column not only shows the syntax elements selected and used, but also all those that can be used for the current NC function.

■ Editing an NC function in 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 editor** mode.

Further information: "Editing NC functions", Page 219

Inserting NC functions

Inserting an NC function directly with keys or buttons

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 by selecting it

To insert a new NC function:

Insert 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.
Paste	▶ Select **Paste**
	> The control creates a new NC block and starts the dialog.
	▶ Follow the instructions in the dialog
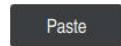
Inserting an NC function in the Text editor mode

To insert an NC function:
- ▶ Enter any character
- > The control inserts an NC block.
- > Depending on the setting of the **Autocomplete in text mode** toggle switch, the control displays a selection menu with possible syntax initiators.
  Further information: "Settings in the Program workspace", Page 206
- ▶ Select the desired syntax initiator
- ▶ Enter the value as needed
- > Depending on the setting of the **Autocomplete in text mode** toggle switch, the control displays a selection menu with possible syntax elements.
- ▶ Select the desired syntax element

Editing NC functions

Editing an NC function in the Klartext editor mode

To edit an NC function in the **Klartext editor** 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	▶ Conclude entry (e.g., by pressing **END**)

Editing an NC function in the Form column
If the **Klartext editor** mode is active, you can also use the **Form** column.

To edit an 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	▶ Complete your input (e.g., with the **Confirm** button)

Editing an NC function in the text editor mode

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 as needed

> The control displays the corresponding error description.

▶ Close the NC block

> The control might open the **NC block auto-correction** window with a solution proposal.

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

---
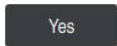
i If you are editing an NC block with syntax errors, the only way to cancel editing is to press the **ESC** key.

Notes

NOTICE

Caution: Data may be lost!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. The changes cannot be undone on the control. This means that any such deletion or altering of data is permanent!

▶ Edit NC programs in the **Program** workspace only

■ 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 213

■ 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 107

■ 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 TNC bar", Page 155

■ 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 currently being executed in the **Program Run** operating mode.

■ In the **Klartext editor** mode, you can insert line breaks within comments or structuring items.

Notes on the Text editor mode

- 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.

  Further information: "Using the Program workspace", Page 211

- In Text editor mode, you can insert line breaks anywhere in your text. If you later edit the NC functions in the **Klartext editor** mode, the control will remove the line breaks after saving. The line breaks will be preserved in comments and structuring items even after editing.
- When you program a cycle using the active auto-complete function, you can select the **Only downwardly-compatible cycle parameters** or **With optional cycle parameters** option.

  When you select **Only downwardly-compatible cycle parameters**, you can add optional cycle parameters later on. For this purpose, insert a line break after the last line.

  Further information: "General information on cycles", Page 222

7.4 Working with cycles

7.4.1 General information on cycles

General information

> ⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
>
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.
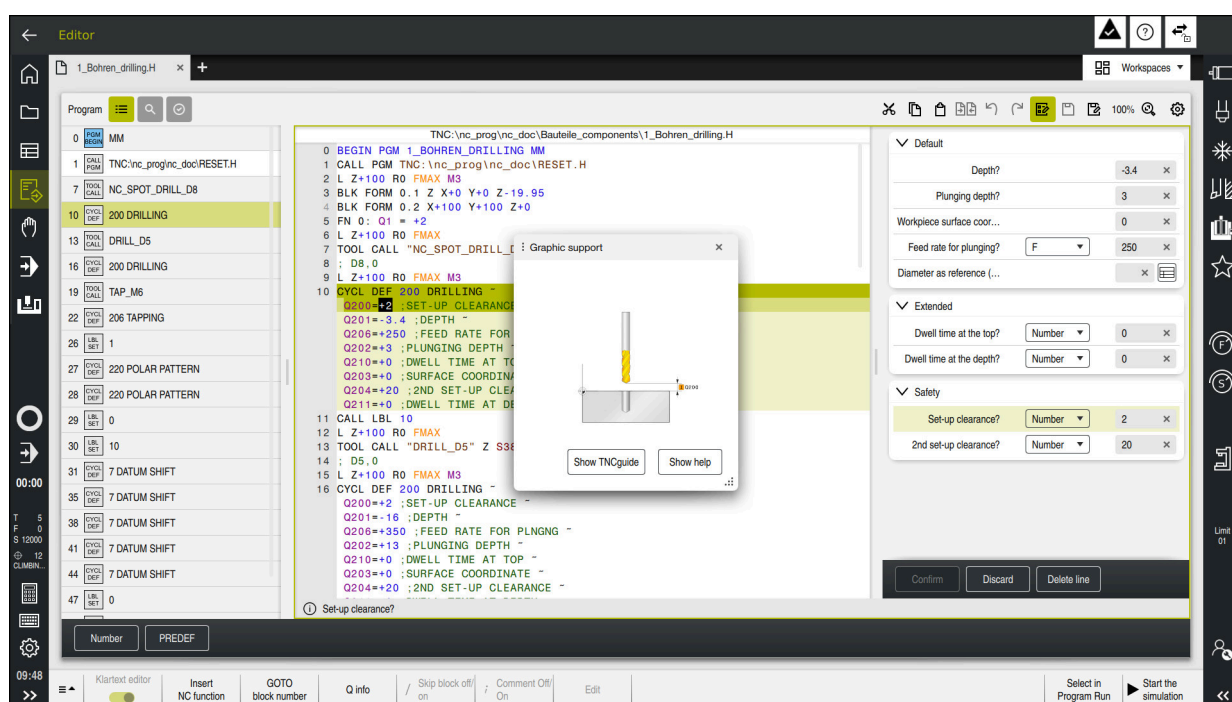
Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations. This greatly simplifies the task of creating programs. The cycles are also useful for frequently recurring machining operations that comprise several working steps. Most cycles use Q parameters 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

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. The section "Titelseite_Neue und geänderte Funktionen" gives you an overview of the optional Q parameters that have been added in this software version. 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 . 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.

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 machining cycles via the CYCL 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.

Inserting touch-probe cycles via the TOUCH PROBE key:

TOUCH PROBE

- ▶ Press 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

> 🛈 For some cycle parameters, the control provides selectable choices via the action bar or the form.
>
> If an input option specifying a defined behavior is stored in certain 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 following options:
> - ■ 0 | Tool tip
> - ■ 1 | Cutting edge corner

Cycle input form

The control provides a **FORM** for various functions and cycles. This **FORM** allows you to enter various syntax elements or cycle parameters.

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.

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.
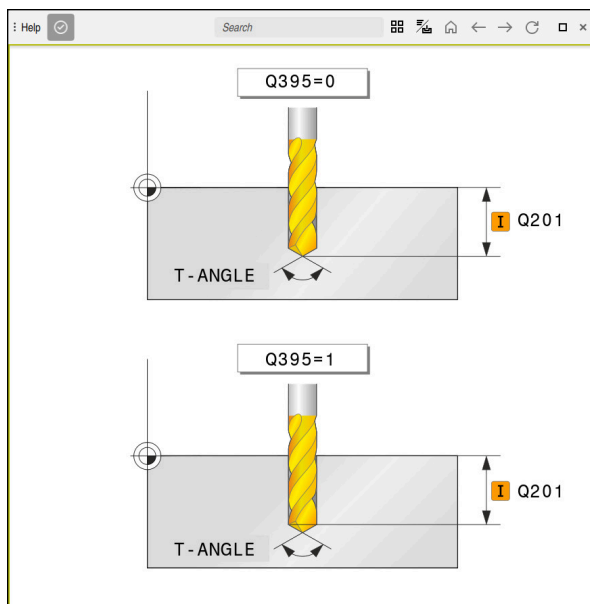
The **Help** workspace with a help graphic for a cycle parameter

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 machining cycle that was defined last 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**)

> **i** 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:

Syntax	Further information
CYCL CALL	Page 227
CYCL CALL PAT	Page 227
CYCL CALL POS	Page 228
M89/M99	Page 228

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined machining cycle once. The starting point of the cycle is the position that was programmed last before the **CYCL CALL** 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 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 394

Further information: "Point tables", Page 391

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 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 machining 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

i 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**:
▶ 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.

To define an NC program as a cycle:

Insert
NC function

▶ Select **Insert NC function**
> The control opens the **Insert NC function** window.

CYC

▶ Select **SEL CYCLE**

▶ Select file name, string parameter or file

To call an NC program as a cycle:

CYCL
CALL

▶ Press the **CYCL CALL** key
> The control opens the **Insert NC function** window.
> or
▶ Program **M99**

ℹ ■ 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 for approaching is calculated based on:
■ the maximum Z position when pattern machining is started
■ 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.

7.4.2   General information about touch probe cycles

Method of function

> ⚙ ■ Refer to your machine manual.
> ■ The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
> ■ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
> ■ If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.
> ■ The control's full range of functions is available only if the **Z** tool axis is used.
> ■ Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.
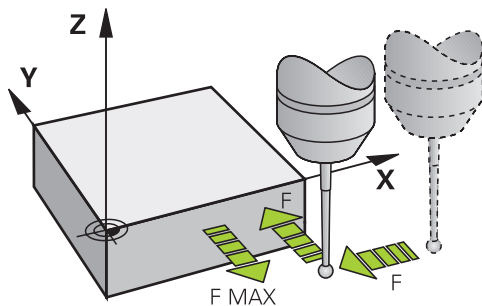
The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Whenever the control runs a touch probe cycle, the 3D touch probe approaches the workpiece parallel to the axis. This is also true during an active basic rotation or with a tilted working plane. The machine manufacturer will determine the probing feed rate in a machine parameter.

Further information: "General information about touch probe cycles", Page 230

When the probe stylus contacts the workpiece,

■ the 3D 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 (#17 / #1-05-1)", Page 1257

- Preset table

  Further information: "Preset table *.pr", Page 1710

- Datum table

  Further information: "Datum table *.d", Page 1721

- Reference systems

  Further information: "Reference systems", Page 690

- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1013

Requirements

- Software option Touch-Probe Functions (#17 / #1-05-1)

- Calibrated workpiece touch probe

  Further information: "Calibrating the workpiece touch probe", Page 1272

Working with an L-shaped stylus

In addition to a **SIMPLE** stylus, probing cycles **444** and **14xx** also support the **L-TYPE** stylus, which is L-shaped. The L-shaped stylus must be calibrated prior to use.

HEIDENHAIN recommends calibrating the stylus with the following cycles:

- Radius calibration: Cycle 460 CALIBRATION OF TS ON A SPHERE (#17 / #1-05-1)

- Length calibration: Cycle 461 TS CALIBRATION OF TOOL LENGTH (#17 / #1-05-1)

Stylus orientation must be permitted via **TRACK ON** in the touch probe table. During the probing process, the control orients the L-shaped stylus to the given probing direction. If the probing direction is identical to the tool axis, then the control orients the touch probe to the calibration angle.

> ℹ️
> - The control does not show the arm of the stylus in the simulation. The arm is the angled part of the L-shaped stylus.
> - The software option **DCM** (#40 / #5-03-1) does not monitor the L-shaped stylus.
> - In order to achieve maximum accuracy, the feed rate during calibration must be identical to the feed rate during probing.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

Notes

> **NOTICE**
>
> **Danger of collision!**
>
> When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
>
> - 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.

General information on the touch-probe table

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
■ Set-up clearance
■ Feed rate for pre-positioning
■ Touch probe orientation
■ Serial number
■ Reaction in case of collision

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application, the control provides touch probe cycles in **Manual** mode 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 (#17 / #1-05-1)", Page 1257

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 functionality, which the control requires in various cycles, always have the same number: For example, **Q260** is always 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.

Further information: "Defining cycles", Page 224

> **ℹ** For the various cycle parameters, the control provides selectable choices via the action bar or the form.

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.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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 touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

Note regarding machine parameters

■ Depending on how the optional machine parameter **chkTiltingAxes** (no. 204600) is set, 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.

Notes in connection with programming and execution

- Please note that the units of measure in the measuring log and in return parameters depend on the setting in 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 apply the inclined position as a workpiece rotation only if a table rotary axis exists on the machine and if its orientation is perpendicular to the workpiece coordinate system **W-CS**.
  Further information: "Comparison of offset and 3D basic rotation", Page 1288

Pre-positioning

Before each probing operation, the control pre-positions the touch probe.

Pre-positioning is done in the inverse probing direction.

The distance between the probing point and the pre-position results from the following values:
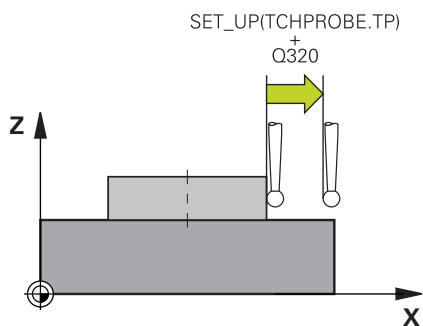
- Ball-tip radius **R**
- **SET_UP** from the touch-probe table
- **Q320 SET-UP CLEARANCE**

Positioning logic

Touch-probe cycles with numbers from **400** through **499** or **1400** through **1499** pre-position the touch probe according to the following positioning logic:

Current position > Q260 CLEARANCE HEIGHT
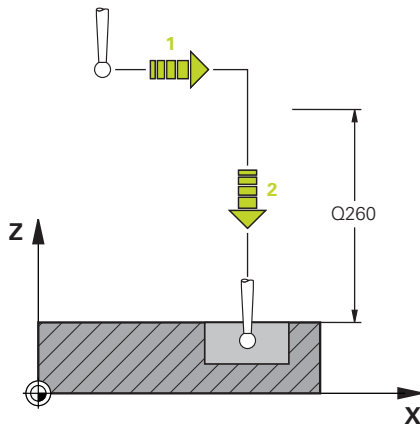
1   The control positions the touch probe at **FMAX** at the pre-position in the working plane.

   Further information: "Pre-positioning ", Page 234

2   Then, the control positions the touch probe at **FMAX** in the tool axis, directly at probing height.

Current position < Q260 CLEARANCE HEIGHT
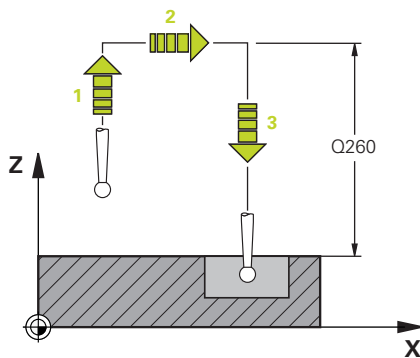
1   The control positions the touch probe at **FMAX** at **Q260 CLEARANCE HEIGHT**.

2   The control positions the touch probe at **FMAX** to the pre-position in the working plane.

   Further information: "Pre-positioning ", Page 234

3   Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.

7.4.3 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

---

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections and thus, undesired behavior. Danger of collision during machining!

- ▶ Only use variable ranges recommended by HEIDENHAIN
- ▶ Do not use pre-assigned variables
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- ▶ Check the machining sequence using the simulation

Further information: "Calling cycles", Page 227
Further information: "Variables: Q, QL, QR and QS parameters", Page 1006

7.4.4 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 448
■ Boring	Page 487
■ Counterboring, centering	
■ Tapping	Page 494
■ Thread milling	Page 510
Pockets/studs/slots	
■ Pocket milling	Page 539
■ Stud milling	Page 565
■ Slot milling	
■ Face milling	Page 661
Coordinate transformations	
■ Mirroring	Page 716
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 586
■ Cylinder surface machining	Page 916
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 626
Point patterns	
■ Bolt hole circle	Page 405
■ Linear hole pattern	
■ Data Matrix code	
Special cycles	
■ Dwell time	Page 898
■ Oriented spindle stop	
■ Tolerance	
■ Program call	Page 368
■ Engraving	Page 681

Measuring cycles

Cycle group	Further information
Rotation	
▪ Probing of plane, edge, two circles, beveled edge	Page 1310
▪ Basic rotation	
▪ Two holes or studs	
▪ Via rotary axis	
▪ Via C-axis	
Preset/Position	
▪ Rectangle, inside or outside	Page 1378
▪ Circle, inside or outside	
▪ Corner, inside or outside	
▪ Center of bolt circle, slot or ridge	
▪ Touch probe axis or single axis	
▪ Four holes	
Measuring	
▪ Angle	Page 1478
▪ Circle, inside or outside	
▪ Rectangle, inside or outside	
▪ Slot or ridge	
▪ Bolt hole circle	
▪ Plane or coordinate	
Special cycles	
▪ Measuring or measuring in 3D	Page 1535
▪ Probing in 3D	Page 1546
▪ Fast probing	
▪ Extrusion probing	
Calibrating the touch probe	
▪ Calibrating the length	Page 1231
▪ Calibration in a ring	
▪ Calibration on a stud	
▪ Calibration on a sphere	

Cycle group	Further information
Measuring kinematics	
■ Saving the kinematics	Page 1575
■ Measure kinematics	
■ Preset compensation	
■ Kinematics grid	
Measuring the tool (TT)	
■ Calibrating the TT	Page 1555
■ Tool length, radius or measuring completely	Page 1249
■ Calibrating the IR-TT	

8

Technology-Specific NC Programming

8.1 Switching the operating mode with FUNCTION MODE

Application

With **FUNCTION MODE SET**, you can activate settings defined by the machine manufacturer, such as changes of the traverse range.

Related topics

- Editing kinematic models in the **Settings** application
  Further information: "Channel Settings", Page 1782

Requirement

- 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.

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.

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

Input

11 FUNCTION MODE SET "Range1"	; Activate the machine manufacturer setting

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **FUNCTION MODE**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION MODE	Syntax initiator for the machining mode
MILL or **SET**	Select the machining mode or machine manufacturer setting
Name or **QS**	Name of a kinematic model or machine-manufacturer setting
	Fixed or variable name
	Selection by means of a selection window
	Optional syntax element

Notes

- 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 functions **Tilt working plane** (#8 / #1-01-1) or **TCPM** (#9 / #4-01-1) are active, you cannot switch the machining mode.

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

- Representation of the workpiece blank in the **Simulation** workspace
  Further information: "The Simulation Workspace", Page 1197

Description of function

You define the blank relative to the workpiece preset.
Further information: "Presets in the machine", Page 196
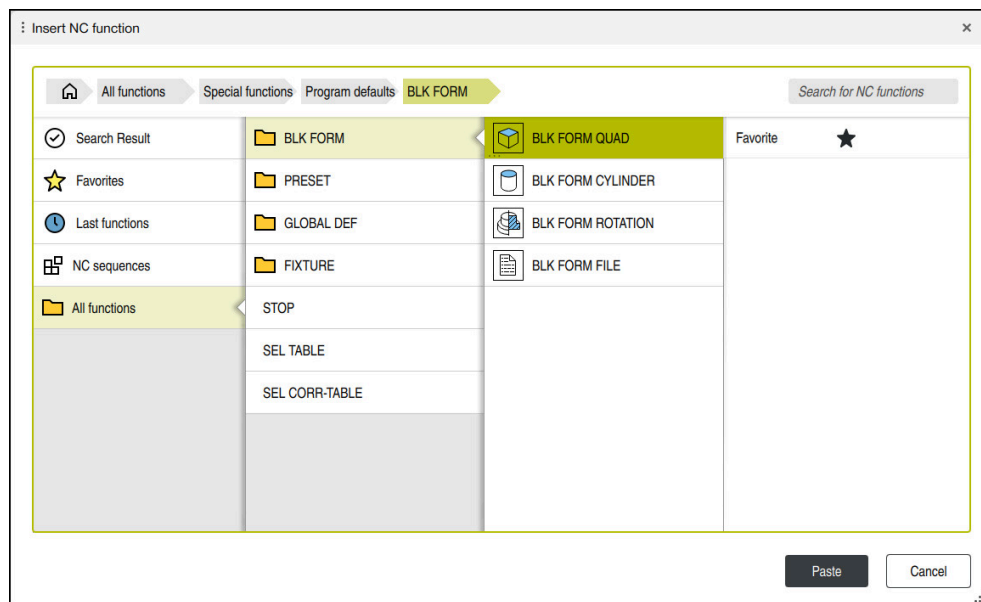
The **Insert NC function** window for workpiece blank definition

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 117
The control offers the following workpiece blank definitions:

Icon	Meaning	Further information
	BLK FORM QUAD Cuboid workpiece blank	Page 246
	BLK FORM CYLINDER Cylindrical workpiece blank	Page 247
	BLK FORM ROTATION Rotationally symmetric blank with a definable contour	Page 248
	BLK FORM FILE STL file as workpiece blank and finished part	Page 249

Notes

NOTICE

Danger of collision!

Even if Dynamic Collision Monitoring (DCM) is active, the control will not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

▶ Activate the **Advanced checks** toggle switch for the simulation
▶ Check the machining sequence using a simulation
▶ Carefully test your NC program or program section in the **Single Block** mode

---

The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

- There are various ways for selecting files or subprograms:
  - Enter the file path
  - Enter the number or name of the subprogram
  - Select the 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 directory as the calling NC program, it might be sufficient to enter just the file name.

- To make the control represent 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 and for the radius.

- The control displays the workpiece blank in the simulation only after having processed the entire workpiece blank definition.

- Even if you have closed the **Insert NC function** window or want to 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. Even if several workpieces are clamped in the machine, the control can monitor only the active workpiece blank!

  Further information: "Advanced checks in the simulation", Page 880

- In the **Simulation** workspace you can export the current workpiece view as an STL file. This function allows you to create missing 3D models, for example semi-finished parts if there are several machining steps.

  Further information: "Exporting a simulated workpiece as STL file", Page 1209

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
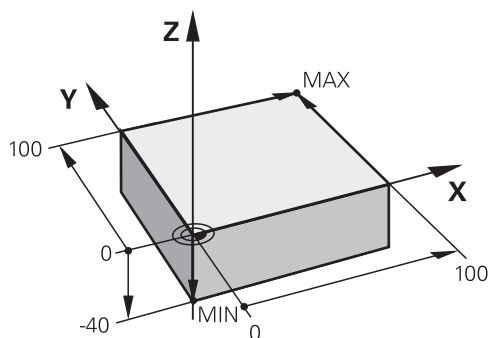
Cuboid workpiece blank with MIN point and MAX point

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 196

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	Start of syntax 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
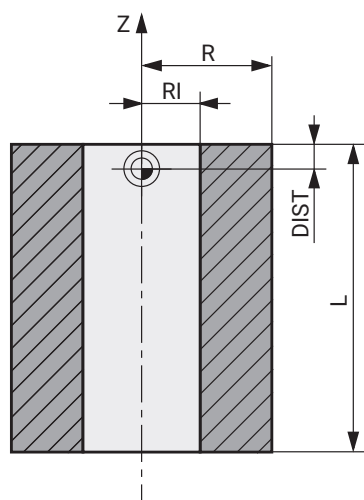
Cylindrical blank

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 |

To navigate to this function:

Insert NC function ▶ Special functions ▶ Program defaults ▶ BLK FORM ▶ BLK FORM CYLINDER

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM CYLINDER	Syntax initiator for cylindrical workpiece blank
Z	Rotary 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
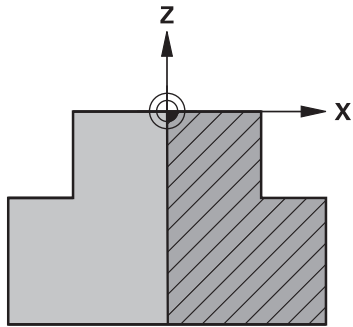
Blank contour with tool axis **Z** and main axis **X**

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

To navigate to this function:

Insert NC function ▶ Special functions ▶ Program defaults ▶ BLK FORM ▶ BLK FORM ROTATION

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM ROTATION	Syntax initiator for rotationally symmetric workpiece blank
Z	Rotary 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 the software option CAD Import (#42 / #1-03-1), you can load contours from CAD files and save them in subprograms or separate NC programs.
  Further information: "Opening CAD files with CAD Viewer", Page 1107

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ BLK FORM ▶ BLK FORM FILE

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM FILE	Syntax initiator for an STL file as workpiece blank
File or **QS**	Path of the STL file
TARGET	STL file as finished part
	Optional syntax element
File or **QS**	Path of the STL file
	Fixed or variable path

Notes

- In the **Simulation** workspace you can export the current workpiece view as an STL file. This function allows you to create missing 3D models, for example semi-finished parts if there are several machining steps.
  Further information: "Exporting a simulated workpiece as STL file", Page 1209
- After integrating a workpiece blank and a finished part, you can compare the models in the simulation and easily identify residual material.
  Further information: "Model comparison", Page 1215
- The control loads binary-format STL files quicker than ASCII-format STL files.
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

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 253
- Further tool data are needed to completely define the tool. Take these tool data from the manufacturer's tool catalog, for example.

  Further information: "Tool data for the tool types", Page 264
- Save all collected tool data of this tool in the tool management.

  Further information: "Tool management ", Page 269
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 273
- After finishing tool definition, program a tool call within an NC program.

  Further information: "Tool call by TOOL CALL", Page 279
- 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 286
- 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 287
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: "Tool radius compensation", Page 798

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 196

10.2.1 Tool carrier reference point
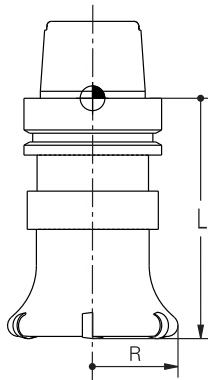
The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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 269

Further information: "Measuring the tool by scratching", Page 1284

10.2.2　Tool tip TIP
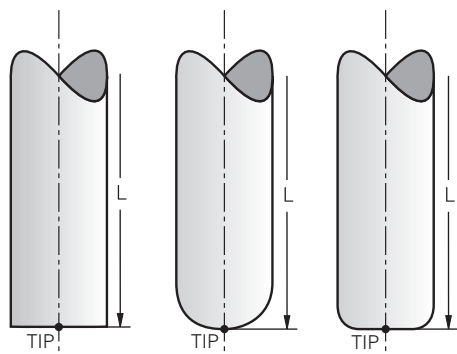
The tool tip has the greatest distance from the tool carrier reference point. The tool tip is the origin of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 703

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**

Further information: "Tool data for the tool types", Page 264

The tool tip is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 255

10.2.3　Tool center point (TCP, tool center point)
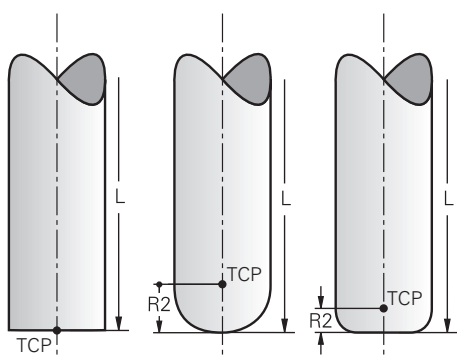
The tool center point is the center of the tool radius **R**. If a second tool radius (**R2**) is defined, the tool center point is offset from the tool tip by this value.

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 264

The tool center point is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 255

10.2.4 Tool location point (TLP, tool location point)
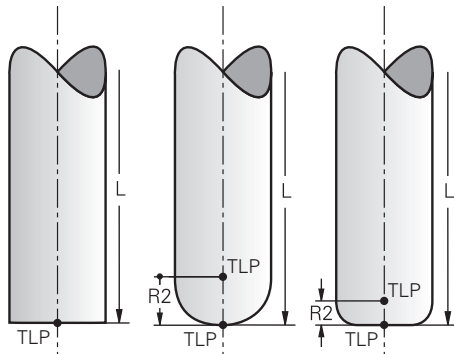
The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

In the function **FUNCTION TCPM** (#9 / #4-01-1), 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 (#9 / #4-01-1)", Page 787

10.2.5 Tool rotation point (TRP, tool rotation point)
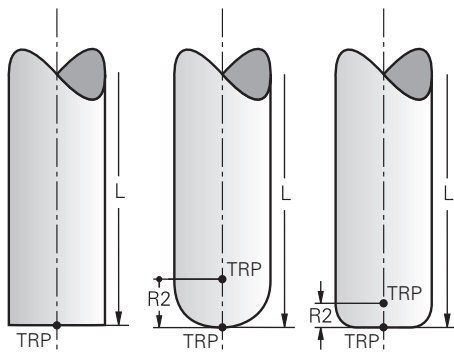
When applying the tilting function with **MOVE** (#8 / #1-01-1), the control tilts the tool about the tool center of rotation. By default, the tool center of rotation is at the tool tip.

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 771

In the function **FUNCTION TCPM** (#9 / #4-01-1), you can also choose the tool center of rotation to be at the tool center point.

10.2.6   Tool radius 2 center (CR2, center R2)
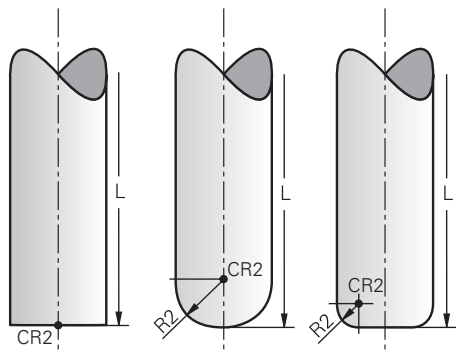
The control uses the tool radius 2 center in conjunction with 3D tool compensation (#9 / #4-01-1). In the case of straight lines **LN**, the surface-normal vector points to that point and defines the direction of the 3D tool compensation.

Further information: "3D tool compensation (#9 / #4-01-1)", Page 805

The tool radius 2 center is offset from the tool tip and the cutting edge by the **R2** value.

The tool radius 2 center is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 255

10.3   Tool data

10.3.1   Tool ID number

Application

Each tool has a unique number which equals the row number of the tool management. Each tool ID number is unique.

Further information: "Tool management ", Page 269

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 279

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.

In conjunction with AFC (#45 / #2-31-1), the following characters are not permitted in the tool name: # $ & , .

Further information: "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

Note

■ Assign unique tool names!

If you define identical tool names for multiple tools, the control will look 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 the least remaining tool life.

10.3.3 Database ID

Application

In a tool database for all machines, you can identify tools with unique database IDs (e.g., within a workshop). This allows you to coordinate the tools of multiple machines more easily.

The database ID is entered in the **DB_ID** column of the tool management.

Related topics

■ **DB_ID** column of tool management

Further information: "Tool table tool.t", Page 1684

Description of function

The database ID is stored in the **DB_ID** column of the tool management.

For indexed tools, you can define the database ID either only for the physically existing main tool or as an ID for the data record at each index.

For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.

Further information: "Indexed tool", Page 258

A database ID may contain a maximum of 40 characters and is unique in the tool management.

The control does not allow a tool call with the database ID.

10.3.4 Indexed tool

Application

Using an indexed tool, several different sets of 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.

Requirement

■ Main tool has been defined

Description of function

Tools with multiple lengths and radii cannot be defined in one row of the tool management table. Additional table rows are required, specifying the full definitions of the indexed tools. The lengths of the indexed tools, starting from the maximum tool length, approach the tool carrier preset as the index increases.

Further information: "Tool carrier reference point", Page 253

Further information: "Creating an indexed tool", Page 259

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 for centering, for example. The indexed tool defines a point along the tool tooth. This can be used for deburring, for example.

■ 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 ●	▶ Enable **Edit**
	> The control enables tool management for editing.
Insert tool	▶ Select **Insert tool**
	> The control opens the **Insert tool** window.
	▶ Select the desired tool type
	▶ Define the tool number of the main tool (e.g., **T5**)
OK	▶ Press **OK**
	> The control adds table row **5**.
	▶ Define any required tool data, including the maximum tool length
	Further information: "Tool data for the tool types", Page 264
Insert tool	▶ Select **Insert tool**
	> The control opens the **Insert tool** pop-up window.
	▶ Enable the **Index** check box
	> The control adds the next free index number for the currently selected tool (e.g., **T5.1**).
OK	▶ Press **OK**
	> The control inserts table row **5.1** with the tool data of the main tool.
	▶ Correct any deviating tool data
	Further information: "Tool data for the tool types", Page 264

> **i** 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 253

Notes

- The control describes some parameters automatically, for example the current tool age **CUR_TIME**. The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 1684

- When you create an indexed tool, the control will copy the tool data from the previous table row. The previous table row may be the main tool or an existing indexed tool.

- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.

- If you delete a main tool, the control will delete all associated indexed tools as well.

- If you copy or cut indexed tools only, you can use **Append** to add the indices to the currently selected tool.

  Further information: "Context menu in the Tables operating mode", Page 1176

- Up to nine indexed tools can be added to each main tool.

- If you define a replacement tool **RT**, this applies to the respective table row exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This means, for example, that the main tool can still be used.

  Further information: "Automatically inserting a replacement tool with M101", Page 999

Example of T-slot milling cutter
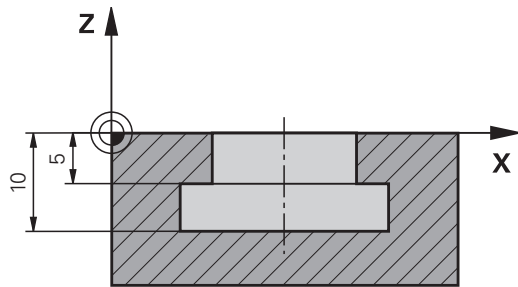
In this example, you program a T-slot with dimensions referring to the top and bottom edges as viewed from the coordinates surface. The height of the T-slot is larger than the length of the cutting edge of the tool used. This requires two steps.

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.

> **i** 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

10.3.5 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 269

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
	Face mill (**MILL_FACE**)	14
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Chamfer mill (**MILL_CHAMFER**)	24
	Side milling cutter (**MILL_SIDE**)	25
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4
	Touch probe (**TCHP**) (#17 / #1-05-1)	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
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19

Icon	Tool type	Number
	Circular thread mill (**ZBGF**)	20

These tool types allow filtering the tools in the tool management.
Further information: "Tool management ", Page 269

10.3.6 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 269
- Tool types
  Further information: "Tool types", Page 261

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 for Tools (#17 / #1-05-1)", Page 1555
- 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 1207
- Machining or touch probe cycles
  Further information: "Cycles for Drilling, Centering and Thread Machining", Page 445
  Further information: "Milling Cycles", Page 535
  Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

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 following milling and drilling tool types: **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 following milling and drilling tool types: **Milling tool** **Roughing mill** **Finishing cutter** **Ball-nose cutter** **Toroid cutter**

Icon and parameter	Meaning	Intended use
 PITCH	Thread pitch	Recommended for the following milling and drilling tool types: ■ **Tapping tools** ■ **Thread mill** ■ **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 following milling and drilling tool types: ■ **Drill** ■ **NC center drill** ■ **Countersink** ■ **Chamfer cutter**
 NMAX	Maximum spindle speed	Optional
R_TIP	Radius at the tip	Recommended for the following milling and drilling tool types: ■ **Face mill** ■ **Countersink** ■ **Chamfer cutter**

> 🛈 ■ All tool types listed in the **TYP** column are milling and drilling tools except for:
>   ■ **Touch probe** (#17 / #1-05-1)
>   **Further information:** "Tool types", Page 261
> ■ The parameters are described in the tool table.
>   **Further information:** "Tool table tool.t", Page 1684

Tool data for touch probes (#17 / #1-05-1)

NOTICE

Danger of collision!

The control cannot protect L-shaped styli from collisions using Dynamic Collision Monitoring DCM (#40 / #5-03-1). When using a touch probe with an L-shaped stylus there is a risk of collision!

▶ Carefully run in the NC program or program section in the **Program Run Single Block** operating mode

▶ Watch out for possible collisions!

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. When selecting **L-TYPE** in the **STYLUS** parameter, **ON** must be selected
REACTION	Trigger **NCSTOP** or **EMERGSTOP** in case of collision	Required
SET_UP	Set-up clearance	Recommended

Icon and parameter	Meaning	Intended use
DIST	Maximum measuring range	Recommended
CAL_OF1	Center offset in the main axis	Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle.
CAL_OF2	Center offset in the secondary axis	Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle.
CAL_ANG	Spindle angle during calibration	Required when **ON** is selected in parameter **TRACK**
STYLUS	Shape of the stylus	Required If you do not define the parameter, the control uses **SIMPLE**

> ℹ️ ▪ You define touch probes by selecting the **Touch probe** tool type in the **TYPE** column and the touch probe model in the **TYPE** column.
> **Further information:** "Tool types", Page 261
> ▪ The parameters are described in the touch probe table.
> **Further information:** "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

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 138
- Table workspace
  Further information: "The Table workspace", Page 1670
- Form workspace
  Further information: "The Form workspace for tables", Page 1676

Description of function

You can define up to 32,767 tools in the tool management; this is the maximum number of available table rows.

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 1684
- Touch-probe table **tchprobe.tp** (#17 / #1-05-1)
  Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

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 1698

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 256

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.
- The 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!
- Make sure to keep the tool table as short and clear as possible so that it does not impair the computing speed of your control. Use a maximum of 10,000 tool entries in tool management. For example, you can delete all unused tool numbers; tool numbers need not be sequential.
- If the cursor is within the **Table** workspace and the **Edit** toggle 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 the controls finds a tool with the entered characters, it selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window.
- The machine manufacturer uses the machine parameter **CfgTableCellLock** (no. 135600) to define whether and in which cases individual table cells are locked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.

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 for the tool database of your CAM system, for example.

Description of function

The control transmits tool data as a CSV file.

Further information: "File types", Page 829

The tool data transfer file is structured as follows:

- The first row contains the tool table column names that are transferred.
- The other rows contain the tool data to be transferred. The order of the data must match the order of the column names in row 1. A period is used as decimal separator.

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**

▶ Enable **Edit**

> The control enables tool management for editing.

▶ Select **Import**

> The control opens a selection window.

▶ Select the desired CSV file
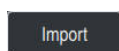
▶ Select **Import**

> 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 as new rows 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
- ▶ Enable **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 1174
- ▶ Select **Mark row**
- ▶ Mark further tools if required
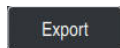
Export
- ▶ Select **Export**
- \> The control opens the **Save as** window.
- ▶ Select a path

> ⓘ   By default, the control saves the transfer file under
>   **TNC:\table**.

- ▶ Enter the file name
- ▶ Select the file type

> ⓘ   You can export the following CSV formats:
>   - **TNC7 (semicolon-separated)**
>   - **iTNC 530 / TNC 640 (comma-separated)**
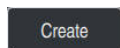
Create
- ▶ Select **Create**
- \> The control will save the file using the selected path.

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 control creates an output of the CSV files with the following formatting:
  - **TNC7 (semicolon-separated)** encloses the values in double quotation marks, the individual values are separated by semicolons
  - **iTNC 530 / TNC 640 (comma-separated)** encloses the values in double curly brackets, the individual values are separated by commas

  Most table calculation programs use the semicolon as the default separator.

  The control is able to import and export data in both formats.

10.5 Tool carrier management

Application

With tool carrier management, you can assign the 3D model of a tool carrier to a tool.

The tool carrier model will be used for the following functions:

■ Representation in the **Simulation** workspace
■ Consideration in Dynamic Collision Monitoring (DCM (#40 / #5-03-1))

Related topics

■ The **Simulation** workspace
  Further information: "The Simulation Workspace", Page 1197
■ Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
■ Adding a tool model to the tool definition (#140 / #5-03-2)
  Further information: "Tool model (#140 / #5-03-2)", Page 277
■ Validating a 3D model for the tool carrier (#56-61 / #3-02-1*)
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804

Requirements

■ Kinematics description
  The machine manufacturer creates the kinematics description
■ Insertion point defined
  The machine manufacturer defines the insertion point for the tool carrier.
■ Tool carrier model exists
  You must save the tool carrier model in the **Toolkinematics** folder.
  Path: **TNC:\system\Toolkinematics**
■ The tool carrier model has been assigned to the tool
  Further information: "Assigning a tool carrier", Page 274

Description of function

The tool carrier model must meet the following requirements:

- Use permitted characters for the file name
  Further information: "Permitted characters", Page 827
- Use a supported format
    - CFG file
    - M3D file
    - STL file
        - Max. 20 000 triangles
        - Triangular mesh forms a closed shell
          Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124

> **i** For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
>
> **Further information:** "Options for fixture files", Page 857

If you are using CFT or CFX files, you must edit the templates in the **ToolHolderWizard** window.

Further information: "Customizing tool carrier templates with ToolHolderWizard", Page 276

10.5.1 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 ⬤	▶ Enable **Edit**
	▶ If applicable, open the **Form** workspace
	▶ In the **Additional geometry data** area, 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 3D model of the tool carrier to the tool.
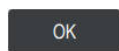
> **i** The tool carrier will only be taken into account after the next tool call.

Notes

- Sample files for tool carrier templates are available on the programming station in the **TNC:\system\Toolkinematics** folder.
- In the simulation, the tool carriers can be checked for collisions with the workpiece.

  Further information: "Advanced checks in the simulation", Page 880
- 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 in the **Z** tool axis. Using the software option Advanced Functions Set 1 (#8 / #1-01-1), you can tilt the working plane to the angle of the removable angled heads and thus keep working with the **Z** tool axis.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM (#40 / #5-03-1)). Thus, the tool carriers are protected against collisions with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

10.6 Customizing tool carrier templates with ToolHolderWizard

> **ℹ** 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 editable dimensions.
>
> They can be downloaded through the following link:
>
> **HEIDENHAIN NC solutions**
>
> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

If you would like to use a CFX or CFT file, you need to parameterize the tool carrier template (i.e., to define the required dimensions). The tool carrier templates can be parametrized in the **ToolHolderWizard** window.

Further information: "Parameterizing tool carrier templates", Page 277

The **ToolHolderWizard** window contains the following icons:

Icon	Meaning
✖	**Close the application**
📂	**Open file**
⬚	Switch between wire frame model and solid object view
⬛	Switch between shaded and transparent view
⊥	Show or hide **Transformation vectors**
ᴬB꜀	Show or hide **Names of collision objects**
⊞	Show or hide **Test points**
⊕	Show or hide **Measuring points**
⁘	**Redo** (restore) the initial view
⬠	**Orientations** (e.g,. top view)

10.6.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

▶ Select the **Files** operating mode

▶ Open the **TNC:\system\Toolkinematics** folder

▶ Double-tap or double-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 **Close the application**

> ⓘ 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!

10.7 Tool model (#140 / #5-03-2)

Application

With the tool model, you can add to a tool definition (e.g., for forward or reverse deburring tools).

The tool model will be used in the following functions only:

- Representation in the **Simulation** workspace
- Consideration in Dynamic Collision Monitoring (DCM (#40 / #5-03-1))

> ⓘ The control will not use the tool model for path contours (e.g., for radius compensation or the **FUNCTION TCPM** function).

Related topics

- The **Simulation** workspace

  Further information: "The Simulation Workspace", Page 1197
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1))

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
- Tool carrier management

  Further information: "Tool carrier management", Page 273
- Validating 3D models with **OPC UA NC Server** (#56-61 / #3-02-1*)

  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804

Requirements

- Software option Dynamic Collision Monitoring (DCM) version 2 (#140 / #5-03-2)
- The tool has been defined in tool management
  Further information: "Tool management ", Page 269
- A suitable tool model exits
  You must save the tool model in the **Toolshapes** folder.
  Path: **TNC:\system\Toolshapes**
  Further information: "Tool model requirements", Page 278
- The tool model has been assigned to the tool
  Further information: "Assigning a tool model", Page 279

Description of function

You can use the tool model for the following tool types:

- Milling tools
- Drilling tools
- Touch probes

Further information: "Tool types", Page 261

Tool model requirements

General requirements

The tool model must meet the following general requirements:

- Use permitted characters for the file name
  Further information: "Permitted characters", Page 827
- Use a supported format
  - M3D file
  - STL file
    - Max. 20 000 triangles
    - Triangular mesh forms a closed shell
      Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124

> ℹ️ For tool models, the same requirements with respect to STL and M3D files apply as for fixtures.
> **Further information:** "Options for fixture files", Page 857

Coordinate system requirements

The coordinate system of the tool model must meet the following requirements:

- The Z axis is the rotary axis of the tool model.
  The control will align the tool model parallel to the tool coordinate system **T-CS**.
  Further information: "Tool coordinate system T-CS", Page 703
- The coordinate origin of the 3D model must be identical to the measured point of the tool. If you measure the tool at the tool tip, you also need to set the coordinate origin of the 3D model to the tool tip.

> ℹ️ If you measured a spherical cutter at the center of the sphere, you need to set the coordinate origin to the center of the sphere as well.

Further information: "Tool tip TIP ", Page 254

10.7.1 Assigning a tool model

To assign a tool model to a tool:

⊞

▶ Select the **Tables** operating mode

▶ Select **Tool management**
▶ Select the tool you want to use

Edit
●

▶ Activate **Edit**

▶ If applicable, open the **Form** workspace
▶ In the **Additional geometry data** area, select the **TSHAPE** parameter
> The control displays the available tool models in the **3D tool model** window.
▶ Select the desired tool model

OK

▶ Select **OK**
> The control assigns the tool model to the tool.

ℹ The tool model will only be taken into account after the next tool call.

Notes

- The control will always take an assigned tool model into account (e.g., for the tool radius **R=0**). The simulation shows the correct shape of the tool model (e.g., in conjunction with a CAM output (center path)).
- When you delete a tool, make sure to remove the tool model from the **Toolshapes** folder as well. This way, you can avoid that the tool model is accidentally referenced for another tool.
- The **LCUTS** column of the tool table is independent of the datum of the tool model. The value is measured from the tool tip of the tool and is effective in the positive Z axis direction.
  Further information: "Tool table tool.t", Page 1684
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

10.8 Tool call

10.8.1 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 999
- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 1684
- Pocket table **tool_p.tch**
  Further information: "Pocket table tool_p.tch", Page 1698

Requirement

- Tool defined
  To call a tool, the tool must be defined in the tool management.
  Further information: "Tool management ", Page 269

Description of function

Upon calling a tool, the control reads the associated row from the tool management. The tool data is displayed on the **Tool** tab of the **Status** workspace.

Further information: "The Tool tab", Page 168

> 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 965

Icons

The NC function **TOOL CALL** offers the following icons:

Icon	Meaning
⤵️	Open selection window for tools
▦	In the **Tool management** application, switch to the selected tool You can change the tool as needed. **Further information:** "Tool management ", Page 269
🖩	Open the **Cutting data calculator** **Further information:** "Cutting data calculator", Page 1181

Input

| 11 TOOL CALL 4 .1 Z S10000 F750 DL +0,2 DR+0,2 DR2+0,2 | ; Call the tool |

To navigate to this function:

Insert NC function ▶ All functions ▶ Tools ▶ TOOL CALL

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL CALL	Syntax initiator for a tool call
Number, **Name**, or **QS**	Tool definition Fixed or variable number or name ℹ 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 282
.1	Step index of the tool Optional syntax element **Further information:** "Input", Page 281
Z	Tool axis By default, tool axis **Z**. 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 282
S or **S(VC =)**	Spindle speed or cutting speed Optional syntax element Selection by means of a selection window **Further information:** "Spindle speed S", Page 283
F, **FZ** or **FU**	Feed rate Alternative feed specifications: feed per tooth or feed per revolution Optional syntax element Selection by means of a selection window **Further information:** "Feed rate F", Page 284
DL	Delta value of tool length Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 796
DR	Delta value of the tool radius Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 796

Syntax element	Meaning
DR2	Delta value of the tool radius 2
	Optional syntax element
	Further information: "Tool compensation for tool length and tool radius", Page 796

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 1684

Tool call for a workpiece touch probe (#17 / #1-05-1)

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 (#17 / #1-05-1)", Page 1694

Updating tool data

A **TOOL CALL** allows updating the data of the active tool even without tool change (e.g., modifying the cutting data or delta values). The tool data that can be modified 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

---

ℹ When a tool number or a tool name or a changed tool axis is programmed in tool call, 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 999

Notes

⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

- 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 286

- 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 tool radius", Page 796

  Further information: "The Positions workspace", Page 149

10.8.2 Cutting data

Application

The cutting data consist of spindle speed **S** or alternatively constant cutting speed **VC** and feed rate **F**.
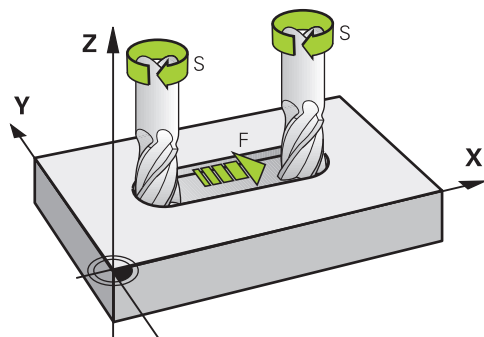
Description of function

Spindle speed S

The spindle speed **S** can be defined in the following ways:

- Tool call with **TOOL CALL**

  Further information: "Tool call by TOOL CALL", Page 279

- **S** button in the **Manual operation** application

  Further information: "The Manual operation application", Page 186

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

Effect

The spindle speed or the cutting speed is active until a new spindle speed or cutting speed is defined in a **TOOL CALL** NC block.

Potentiometers

The speed potentiometer allows varying the spindle speed between 0% and 150% while the program is running. The speed potentiometer setting is active only for machines with infinitely variable spindle drive. The maximum spindle speed depends on the machine.

Further information: "Potentiometers", Page 105

Status displays

The control displays the current spindle speed in the following workspaces:
- The **Positions** workspace
  Further information: "The Positions workspace", Page 149
- The **POS** tab of the **Status** workspace
  Further information: "POS tab", Page 164

Feed rate F

The feed rate **F** can be defined in the following ways:
- Tool call with **TOOL CALL**
  Further information: "Tool call by TOOL CALL", Page 279
- Positioning block
  Further information: "Path Functions", Page 291
- **F** button in the **Manual operation** application
  Further information: "The Manual operation application", Page 186

The feed rate for linear axes is defined in millimeters per minute (mm/min).

The feed rate for rotary axes 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 269

- Feed rate per revolution **FU** in mm/rev

  FU defines the path in millimeters that the tool covers per spindle revolution.

The feed rate defined in a **TOOL CALL** can be called up within the NC program, using **F AUTO**.

Further information: "F AUTO", Page 284

The feed rate defined in the NC program is active up to the NC block in which a new feed rate is programmed.

F MAX

If you define **F MAX**, the control moves at rapid traverse. **F MAX** is non-modal, i.e., it is active only in the block where it is called. Starting with the subsequent NC block, the last previously defined feed rate is active again. The maximum feed rate depends on the machine and may depend on the axis.

Further information: "Feed rate limit F LIMIT", Page 1642

F AUTO

If you defined a feed rate in a **TOOL CALL** block, this feed rate can be used in the next positioning blocks, using **F AUTO**.

F button in the Manual operation application

- If you enter F=0, then the feed rate that the machine manufacturer has defined as minimum feed rate is active
- If the feed rate you entered exceeds the maximum value that has been defined by the machine manufacturer, then the value defined by the machine manufacturer is active

Further information: "The Manual operation application", Page 186

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 active 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 105

Status displays

The control displays the current feed rate in mm/min in the following workspaces:

- The **Positions** workspace

Further information: "The Positions workspace", Page 149

- The **POS** tab of the **Status** workspace

> ⓘ In the **Manual operation** application, the control displays the feed rate with decimal places on the **POS** tab. The control displays the feed rate with a total of six decimal places.

Further information: "POS tab", Page 164

- The control displays the contouring feed rate as follows:
  - If **3D ROT** is active, the contouring feed rate is displayed if multiple axes are moving
  - If **3D ROT** is inactive, the feed-rate display remains empty when more than one axis is moved simultaneously
  - If a handwheel is active, the control shows the contouring feed rate during program run.

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

Notes

- In inch programs, the feed rate must be defined in 1/10 inch/min.
- Make sure to program rapid traverse movements exclusively with the **FMAX** NC function instead of entering extremely high numerical values. This is the only way to ensure that rapid traverse is active on a block-by-block basis and that you can control rapid traverse independently of the machining feed rate.
- When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.

10.8.3 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

To navigate to this function:

Insert NC function ▶ All functions ▶ Tools ▶ TOOL DEF

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL DEF	Syntax initiator for tool pre-selection
Number, **Name**, or **QS**	Tool definition Fixed or variable number or name Selection by means of a selection window ℹ 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 Optional syntax element **Further information:** "Indexed tool", Page 258

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

10.9 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 1702
- Tool usage test in Batch Process Manager (#154 / #2-05-1)

  Further information: "Batch Process Manager (#154 / #2-05-1)", Page 1625

Requirements

- To perform a tool usage test, you need a tool usage file

  In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.

  Further information: "Tool usage file", Page 1702
- The **Generate tool-usage file** setting is set to **Once** or **Always**

  Further information: "Channel Settings", Page 1782
- Use the same tool table for the simulation as for the program run

  Further information: "The Simulation Workspace", Page 1197

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 the **Refresh** icon in the **Tool usage** area of the **Tool check** column

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 1702

The Tool check column in the Program workspace
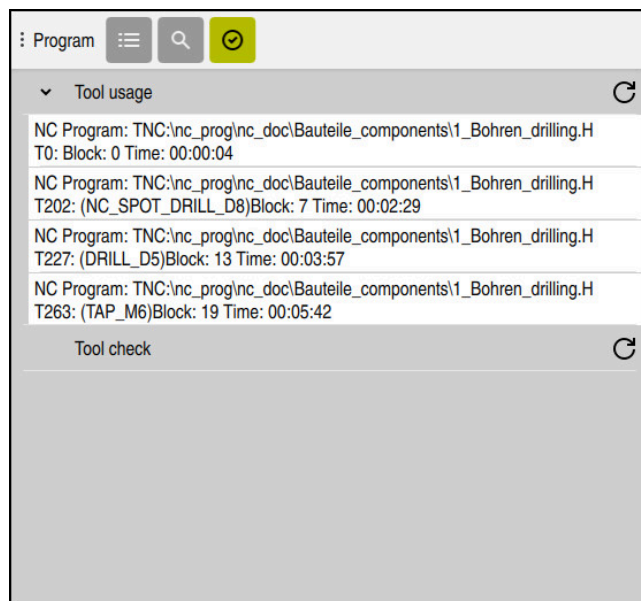
The **Tool check** column in the **Program** workspace

In the **Tool check** column of the **Program** workspace, the control displays the following areas:

- **Tool usage**
  Further information: "The Tool usage area", Page 288
- **Tool check**
  Further information: "The Tool check area", Page 289
- **Perform conditional stop**
  Further information: "Override Controller", Page 1756

Further information: "The Program workspace", Page 203

The Tool usage area

If no tool-usage file has been created yet, the **Tool usage** area is empty.

Further information: "Creating the tool usage file", Page 287

Further information: "Tool usage file", Page 1702

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

Select the **Refresh** icon to create a tool-usage file for your NC program.

The Tool check area

The **Tool check** area is empty until you perform a tool usage test with the **Refresh** icon.

Further information: "Performing the tool usage test", Page 289

When performing the tool usage test, the control checks the following:
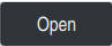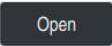
■ The tool is defined in the tool management

Further information: "Tool management ", Page 269

■ The tool is defined in the pocket table

Further information: "Pocket table tool_p.tch", Page 1698

■ 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 1684

Further information: "Tool usage file", Page 1702

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 279

Further information: "Automatically inserting a replacement tool with M101", Page 999

10.9.1 Performing the tool usage test

To perform a tool usage test:

▶ Select the **Editor** operating mode

▶ Select **Add**

▶ Select the desired NC program

▶ Select **Open**

> The control opens the NC program in a new tab.

▶ Open the **Tool check** column

▶ In the **Tool usage** area, select **Refresh**

> The control generates a tool usage file and displays the tools used in the **Tool usage** area.

Further information: "Tool usage file", Page 1702

▶ In the **Tool check** area, select **Refresh**

> The control performs the tool usage test.

> The **Tool check** area shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you double-tap or double-click a tool entry in the **Tool usage** or **Tool check** areas, the control switches to the tool selected in tool management. You can make modifications as needed.

- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.
  Further information: "The Simulation Workspace", Page 1197

- The control saves the tool usage file as a dependent file (***.dep**).
  Further information: "Tool usage file", Page 1702

- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.
  Further information: "Areas of file management", Page 825

- The control displays the order of tool calls of the currently running NC program in the **T usage order** (#93 / #2-03-1) table.
  Further information: "T usage order (#93 / #2-03-1)", Page 1704

- An overview of all tool calls of the NC program active in the program run is displayed by the control in the **Tooling list** table (#93 / #2-03-1).
  Further information: "Tooling list (#93 / #2-03-1)", Page 1706

- The function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.

- The function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. After **IDX** you define the pallet table row.

- 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.

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.
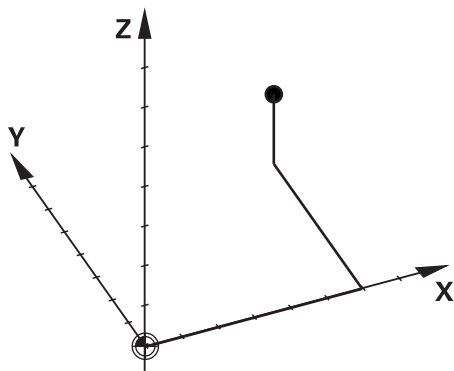
With Cartesian coordinates you can uniquely specify a point in space by defining the three axis values.

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.
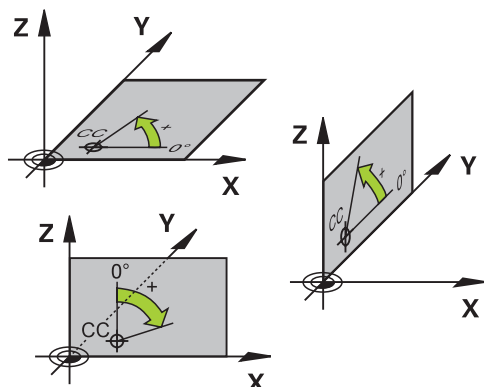
Description of function

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
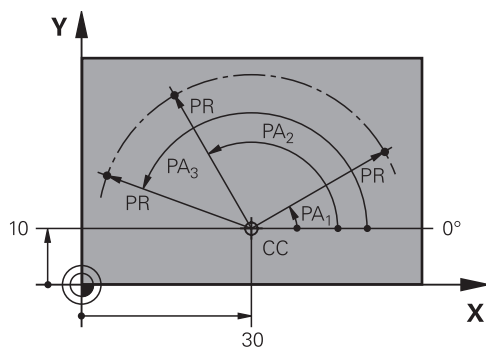
You define the pole **CC** with Cartesian coordinates in two axes. These axes specify the plane and the angle reference axis.

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 input

Application

Absolute input always references 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 values define the target point for positioning.
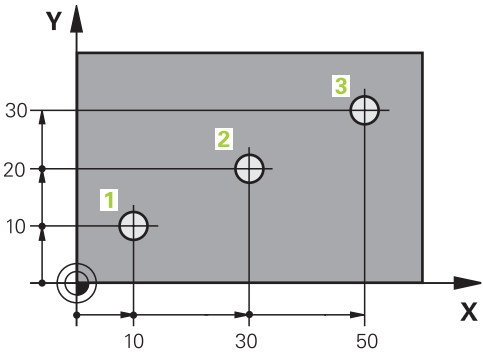
11 L X+10 Y+10 RL F200 M3	; Position at point 1
12 L X+30 Y+20	; Position at point 2
13 L X+50 Y+30	; Position at point 3
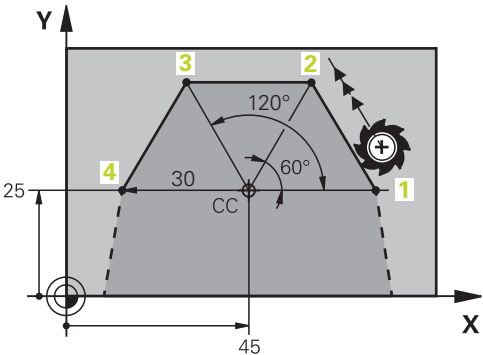
11 CC X+45 Y+25	; Define the pole with two axes using Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position at point 1
13 LP PA+60	; Position at point 2
14 LP PA+120	; Position at point 3
15 LP PA+180	; Position at point 4

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.
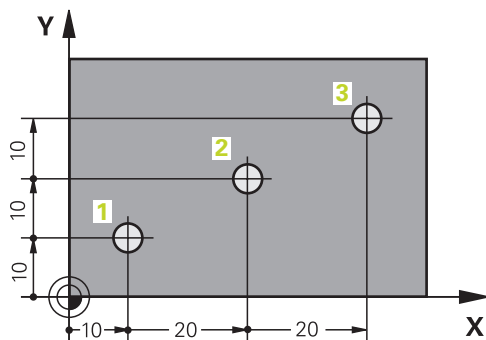
11 L X+10 Y+10 RL F200 M3	; Position to point 1 absolutely
12 L IX+20 IY+10	; Position to point 2 incrementally
13 L IX+20 IY+10	; Position to point 3 incrementally
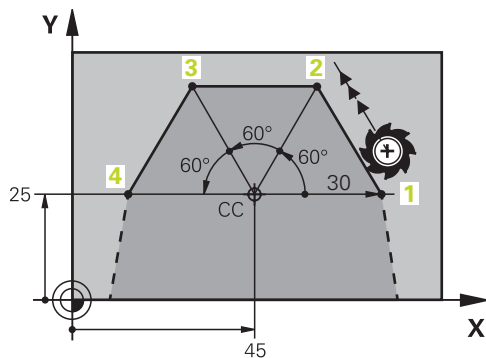
11 CC X+45 Y+25	; Define the pole absolutely in two axes with Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position to point 1 absolutely
13 LP IPA+60	; Position to point 2 incrementally
14 LP IPA+60	; Position to point 3 incrementally
15 LP IPA+60	; Position to point 4 incrementally

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.

> **i** 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
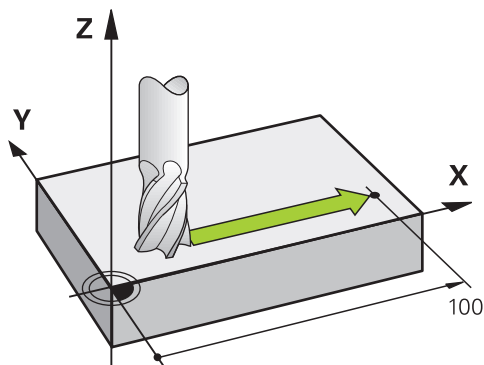
If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Example

L X+100

The tool retains the Y and Z coordinates and moves to the position **X+100**.

Motion in two axes
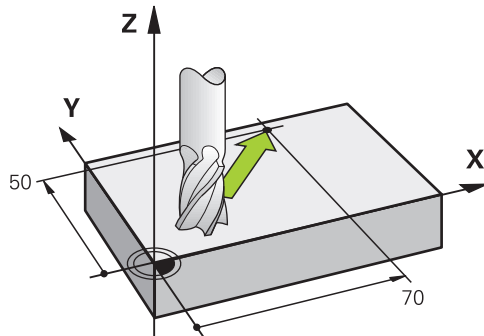
If the NC block contains two coordinates, the control moves the tool in the programmed plane.

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 of milling machines", Page 194

Motion in more than two axes
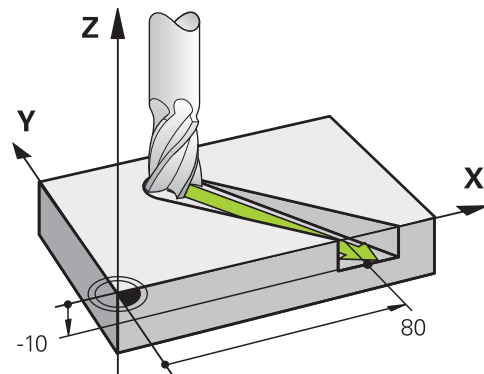
If the NC block contains three coordinate entries, the control moves the tool spatially to the programmed position.

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

> ℹ️ The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.
> If the axis position does not change, you can nevertheless program more than four axes.

Circles and arcs
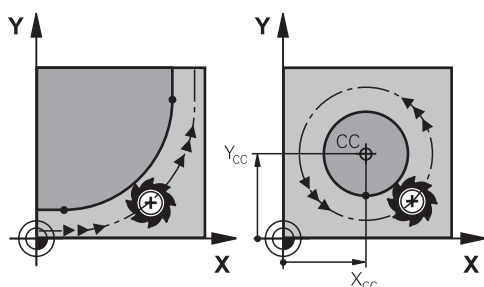
Use the path functions for circular arcs to program circular motions in the working plane.

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
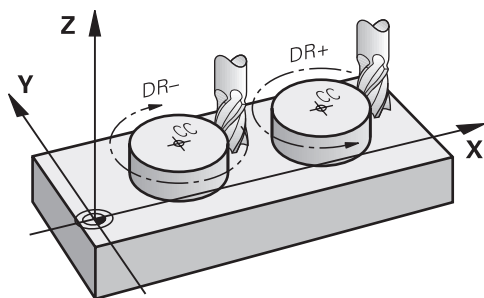
When a circular path has no tangential transition to another contour element, define the direction of rotation as follows:

- 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 798

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 300
CHF	Chamfer **CHF** (chamfer) Chamfer between two straight lines	Page 302
RND	Rounding **RND** (rounding of corner) Circular arc with tangential connection to the preceding and subsequent contour elements	Page 304
CC	Circle center point **CC** (circle center)	Page 306
C	Circular path **C** (circle) Circular path around a circle center **CC** to an end point	Page 308
CR	Circular path **CR** (circle by radius) Circular path with a specified radius	Page 310
CT	Circular path **CT** (circle tangential) Circular path with tangential connection to the preceding contour element	Page 313

11.3.2 Straight line L

Application

With a straight line **L** you program a straight traverse motion in any direction.

Related topics

- Programming a straight line with polar coordinates
  Further information: "Straight line LP", Page 320

Description of function
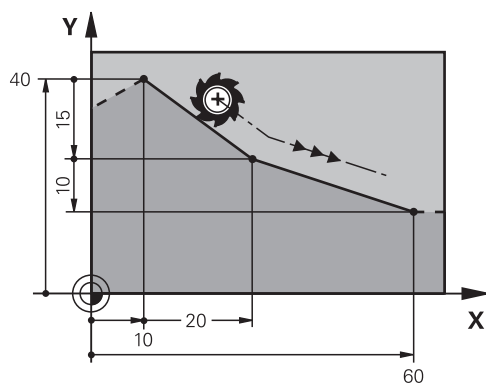
The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.

Input

11 L X+50 Y+50 R0 FMAX M3	; Straight line without radius compensation in rapid traverse

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **L**

The NC function includes the following syntax elements:

Syntax element	Meaning
L	Syntax initiator for a straight line
X, **Y**, **Z**, **A**, **B**, **C,** **U**, **V**, **W**	End point of the straight line as a fixed or variable number Entry: absolute or incremental Optional syntax element
&X, **&Y**, **&Z**	End point of the straight line in a main axis that is deselected with **PARAXMODE** as a fixed or variable number **Further information:** "Select three linear axes for machining with FUNCTION PARAXMODE", Page 938 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Notes

- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.
  Further information: "The Form column in the Program workspace", Page 214
- The **actual position capture** key allows you to program a straight line **L** with all axis values. The values are equivalent to the **Actual pos. (ACT)** mode of the position display.
  Further information: "Position displays", Page 172

Example

11 L Z+100 R0 FMAX M3
12 L X+10 Y+40 RL F200
13 L IX+20 IY-15
14 L X+60 IY-10

11.3.3 Chamfer CHF

Application

The **CHF** chamfer function allows you to insert a chamfer between two straight lines. The size of the chamfer is based on the intersection that you have programmed with the 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
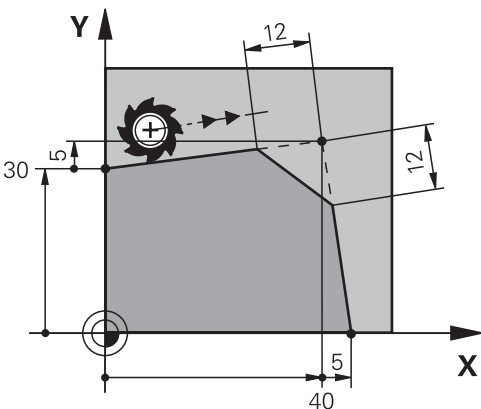
Cutting two straight lines creates contour corners. You can insert a chamfer at these contour corners. The angle of the corner is irrelevant; you simply define the length by which each straight line is shortened. The control does not traverse to the corner point.

If you program a feed rate in the **CHF** block, then this feed rate is in effect only while cutting the chamfer.

Input

| **11 CHF 1 F200** | ; Chamfer with a size of 1 mm |

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **CHF**

The NC function includes the following syntax elements:

Syntax element	Meaning
CHF	Syntax initiator for a chamfer
1	Chamfer size
	Fixed or variable number
F, FAUTO	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element

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

The **RND** rounding arc function allows you to insert a rounding arc between two straight lines. The rounding arc is based on the intersection that you have programmed with the straight lines.

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
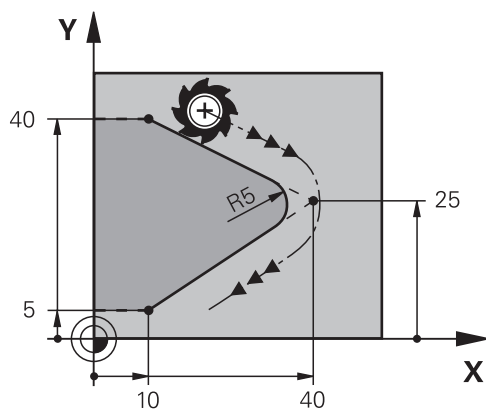
You program the rounding arc between two path functions. The circular arc connects tangentially to the previous and subsequent contour element. The control does not traverse to the intersection.

If you program a feed rate in the **RND** block, then this feed rate is in effect only while cutting the rounding arc.

Input

| **11 RND R3 F200** | ; Radius with a size of 3 mm |

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ RND

The NC function includes the following syntax elements:

Syntax element	Meaning
RND	Syntax initiator for a radius
R	Radius size
	Fixed or variable number
F, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element

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

The **CC** circle center function allows you to define a position as a circle center.

Related topics

■ Programming a pole as a reference point for polar coordinates
Further information: "Polar coordinate datum at pole CC", Page 319

Description of function
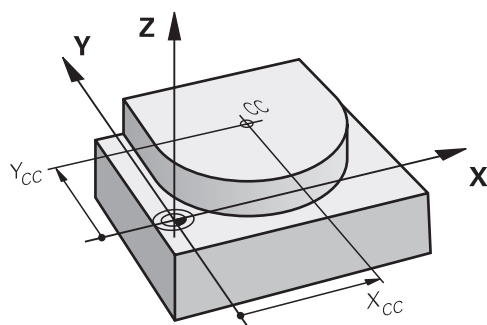
You define a circle center point by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The circle center point remains active until you define a new circle center point. The control does not traverse to the circle center point.

You need to define a circle center point before you can program a circular path with **C**.

ℹ️ The control simultaneously uses the **CC** function as the pole for polar coordinates.
Further information: "Polar coordinate datum at pole CC", Page 319

Input

11 CC X+0 Y+0	; Circle center

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CC

The NC function includes the following syntax elements:

Syntax element	Meaning
CC	Syntax initiator for a circle center
X, **Y**, **Z**, **U**, **V**, **W**	Coordinates of the circle center Fixed or variable number Entry: absolute or incremental Optional syntax element

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.

Related topics

- Programming a circular path with polar coordinates

  Further information: "Circular path CP around pole CC", Page 323

Requirement

- Circle center point **CC** is defined

  Further information: "Circle center point CC", Page 306
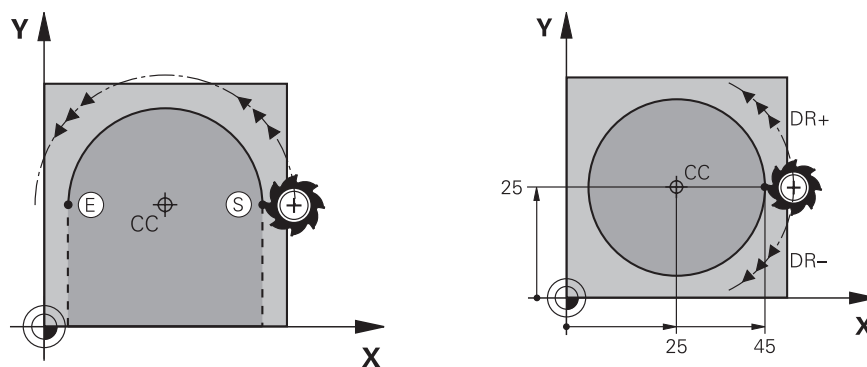
Description of function

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.

> ℹ 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**)

Input

11 C X+50 Y+50 LIN_Z-3 DR- RL F250 M3	; Circular path with linear Z-axis superimpositioning

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ C

The NC function includes the following syntax elements:

Syntax element	Meaning
C	Syntax initiator for a circular path around a circle center
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	End point of the circular path Fixed or variable number Entry: absolute or incremental Optional syntax element
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimposition Fixed or variable number Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 315 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

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.
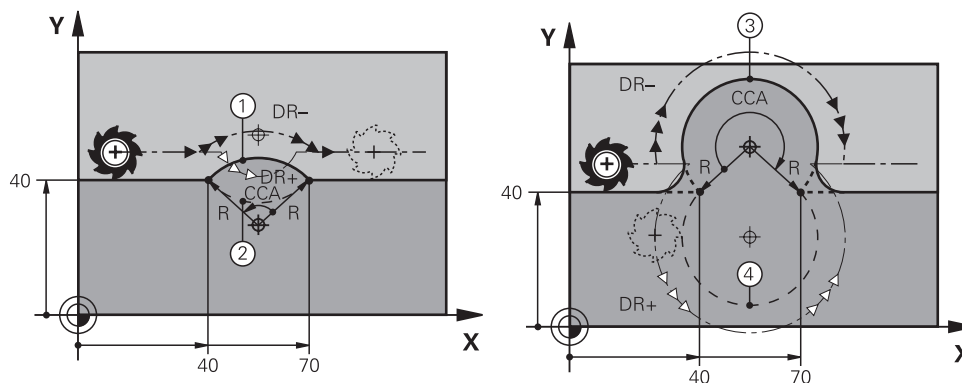
The starting and end points can be connected with four different circular paths of the same radius. The correct circular path is defined with the **CCA** center angle of the circular path radius **R** and the direction of rotation **DR**.

The algebraic sign of the circular path radius **R** is decisive for whether the control selects a center angle that is greater than or less than 180°.

The radius has the following effects on the center 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**)

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
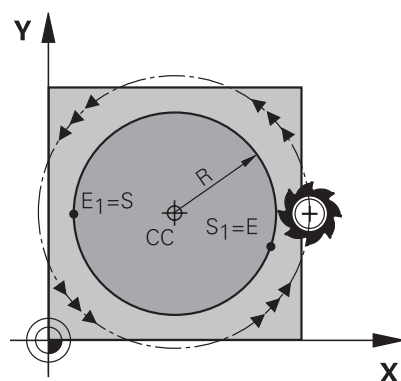
For a full circle, program two circular paths in succession. The end point of the first circular path is the starting point of the second. The end point of the second circular path is the starting point of the first.

Input

11 CR X+50 Y+50 R+25 LIN_Z-2 DR- RL F250 M3	; Circular path with linear Z-axis superimpositioning

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CR

The NC function includes the following syntax elements:

Syntax element	Meaning
CR	Syntax initiator for a circular path with a radius
X, **Y**, **Z**, **A**, **B**, **C,** **U**, **V**, **W**	End point of the circular path Entry: absolute or incremental Optional syntax element
R	Radius of the circular path as a fixed or variable number
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimposition Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 315 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The distance between the starting and end points must not be greater than the circle diameter.

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.

Related topics

■ Programming a tangential connecting circular path with polar coordinates

Further information: "Circular path CTP", Page 325

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
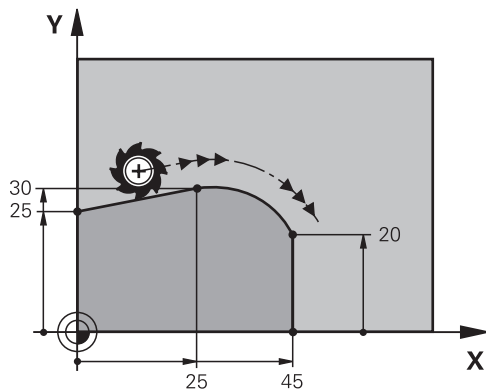
The control moves the tool on a circular path, with a tangential connection, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

When contour elements uniformly merge into another without kinks, then this transition is referred to as tangential.

Input

| 11 CT X+50 Y+50 LIN_Z-2 RL F250 M3 | ; Circular path with linear Z-axis superimpositioning |

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CT

The NC function includes the following syntax elements:

Syntax element	Meaning
CT	Syntax initiator for a circular path with a tangential connection
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	End point of the circular path Entry: absolute or incremental Optional syntax element
LIN_X, **LIN_Y**, **LIN_Z**, **LIN_A**, **LIN_B**, **LIN_C**, **LIN_U**, **LIN_V** or **LIN_W**	Axis and value of the linear superimposition Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 315 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

- The contour element and the circular path should contain both coordinates of the plane in which the circular path is executed.
- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.
  Further information: "The Form column in the Program workspace", Page 214

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 Linear superimpositioning of a circular path

Application

You can linearly superimpose a movement programmed in the working plane, thereby creating a spatial movement.

If, for example, you superimpose a circular path, you create a helix. A helix is a cylindrical spiral, such as a thread.

Related topics

- Linear superimpositioning of a circular path that is programmed with polar coordinates

  Further information: "Linear superimpositioning of a circular path", Page 327

Description of function

You can linearly superimpose the following circular paths:

- Circular contour **C**

  Further information: "Circular path C ", Page 308

- Circular contour **CR**

  Further information: "Circular path CR", Page 310

- Circular contour **CT**

  Further information: "Circular path CT", Page 313

> **i** The tangential transition of the circular path **CT** has an effect only in the axes of the circular plane and not additionally on the linear superimpositioning.

In order to superimpose a linear movement onto circular paths with Cartesian coordinates, additionally program the optional syntax element **LIN**. You can define a main axis, rotary axis or parallel axis (e.g., **LIN_Z**).

Notes

- You can hide the **LIN** syntax element via the settings in the **Program** workspace.

  Further information: "Settings in the Program workspace", Page 206

- Alternatively, you can also superimpose linear movements with a third axis, thereby creating a ramp. A ramp allows you, for example, to plunge into the material with a tool that is not a center-cut tool.

  Further information: "Straight line L", Page 300

Example

A program section repeat allows you to program a helix with the syntax element **LIN**.

This example shows an M8 thread with a depth of 10 mm.

The thread pitch is 1.25 mm. Thus, for a depth of 10 mm, eight thread grooves are required. An initial thread groove is also programmed as an approach path.

11 L Z+1.25 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 LBL 1	
15 C X+4 Y+0 ILIN_Z-1.25 DR-	; Cut the first thread groove
16 LBL CALL 1 REP 8	; Mill the following eight thread grooves, **REP 8** = Number of remaining machining operations

This solution directly uses the thread pitch as the incremental infeed depth per revolution.

REP shows the number of repetitions required for reaching the calculated ten infeed runs.

Further information: "Subprograms and program section repeats with the label LBL", Page 360

11.3.10 Circular path in another plane

Application

You can also program circular paths that do not lie in the active working plane.

Description of function
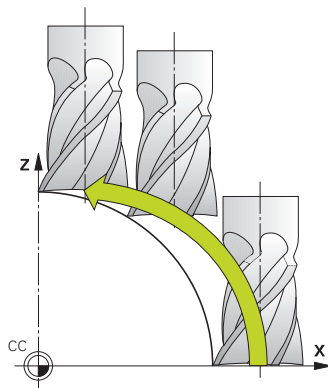
You program circular paths that lie in another plane by entering one axis of the working plane and the tool axis.

Further information: "Designation of the axes of milling machines", Page 194

You can program circular paths that lie in another plane with the following functions:

- **C**
- **CR**
- **CT**

> **i** 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+

11.3.11 Example: Cartesian path functions
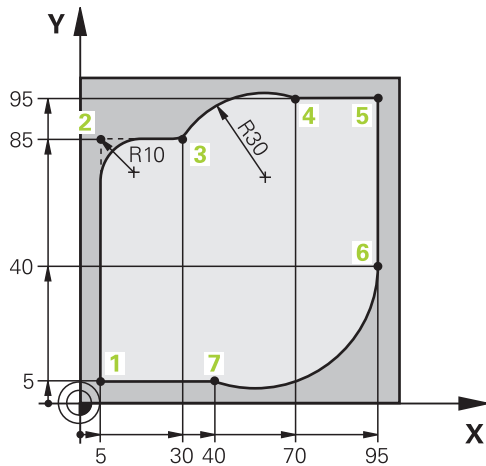
0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Define the workpiece blank for workpiece simulation
3 TOOL CALL 1 Z S4000	; Call the tool in the tool axis and with the spindle speed
4 L Z+250 R0 FMAX	; Retract the tool in the tool axis at rapid traverse FMAX
5 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 320
C	**+**	P	Circular path **CP** (circle polar) Circular path around circle center point or pole **CC** to arc end point	Page 323
CT	**+**	P	Circular path **CTP** (circle tangential polar) Circular path with tangential connection to the preceding contour element	Page 325
C	**+**	P	Helix with circular path **CP** (circle polar) Combination of a circular and a linear motion	Page 327

11.4.2 Polar coordinate datum at pole CC

Application

You must define a **CC** pole before programming with polar coordinates. All polar coordinates are relative to the pole.

Related topics

■ Programming a circle center as a reference point for a circular path **C**
  Further information: "Circle center point CC", Page 306

Description of function
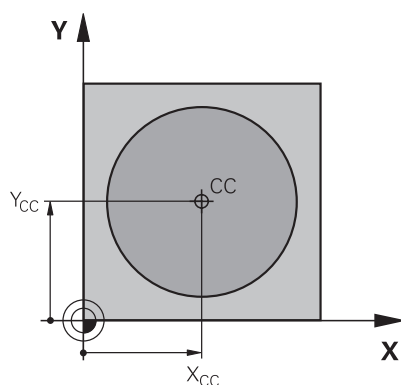
You use the **CC** function to define a position as the pole. You define a pole by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The pole remains active until you define a new pole. The control does not traverse to this position.

Input

11 CC X+0 Y+0	; Pole

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CC

The NC function includes the following syntax elements:

Syntax element	Meaning
CC	Syntax initiator for a pole
X, **Y**, **Z**, **U**, **V**, **W**	Coordinates of the pole
	Fixed or variable number
	Entry: absolute or incremental
	Optional syntax element

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.

Related topics

- Programming a straight line with Cartesian coordinates
  Further information: "Straight line L", Page 300

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 319

Description of function
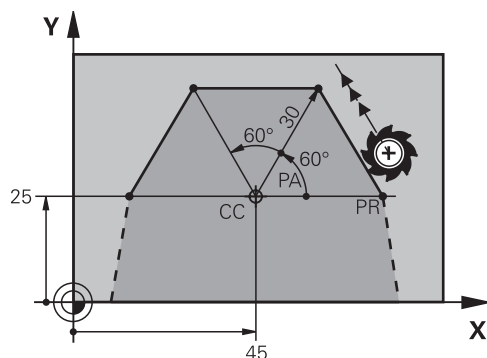
The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

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

Input

11 LP PR+50 PA+0 R0 FMAX M3	; Straight line without radius compensation in rapid traverse

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ L

The NC function includes the following syntax elements:

Syntax element	Meaning
LP	Syntax initiator for a straight line with polar coordinates
PR	Polar coordinate radius
	Fixed or variable number
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Fixed or variable number
	Entry: absolute or incremental
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 798
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

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.

Related topics

- Programming a circular path with Cartesian coordinates

  Further information: "Circular path C ", Page 308

Requirement

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 319

Description of function
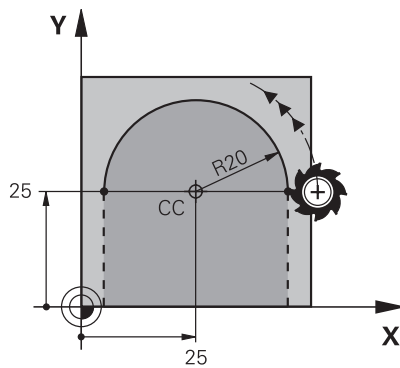
The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block.

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.

Input

| 11 CP PA+50 Z-2 DR- RL F250 M3 | ; Circular path |

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ C

The NC function includes the following syntax elements:

Syntax element	Meaning
CP	Syntax initiator for a circular path around a pole
PA	Polar coordinate angle
	Entry: absolute or incremental
	Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	Axis and value of the linear superimposition
	Entry: absolute or incremental
	Further information: "Linear superimpositioning of a circular path", Page 327
	Optional syntax element
DR	Rotational direction of the arc
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 798
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Notes

- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.
- 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.

Related topics

- Programming a tangentially connecting circular path with Cartesian coordinates

  Further information: "Circular path CT", Page 313

Requirements

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 319

- 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.

Input

11 CTP PR+30 PA+50 Z-2 DR- RL F250 M3	; Circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ CT

The NC function includes the following syntax elements:

Syntax element	Meaning
CTP	Syntax initiator for a circular path with a tangential connection
PR	Polar coordinate radius
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Entry: absolute or incremental
	Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	Axis and value of the linear superimposition
	Entry: absolute or incremental
	Further information: "Linear superimpositioning of a circular path", Page 327
	Optional syntax element
DR	Rotational direction of the arc
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 798
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Notes

- The pole is **not** the center of the contour circle!
- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.
  Further information: "The Form column in the Program workspace", Page 214

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 Linear superimpositioning of a circular path

Application

You can linearly superimpose a movement programmed in the working plane, thereby creating a spatial movement.

If, for example, you superimpose a circular path, you create a helix. A helix is a cylindrical spiral, such as a thread.

Related topics

- Linear superimpositioning of a circular path that is programmed with Cartesian coordinates

   Further information: "Linear superimpositioning of a circular path", Page 315

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 323

Description of function
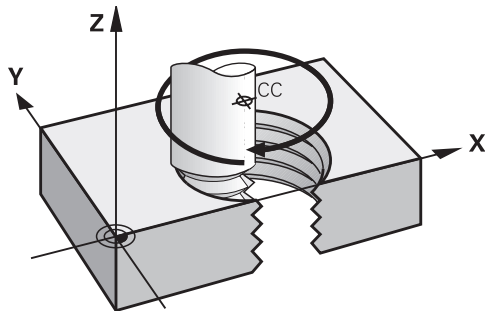
A helix is a combination of a circular path **CP** and a linear motion perpendicular to this path. You program the circular path **CP** in the working plane.

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
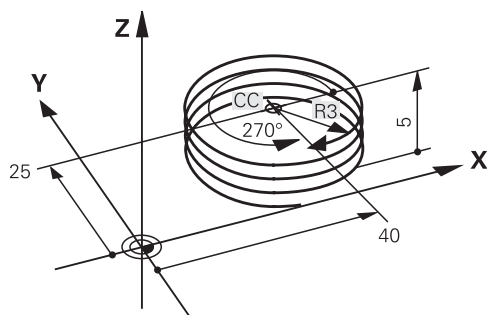
> **ⓘ** Define the same algebraic sign for the direction of rotation **DR** and the incremental total angle **IPA**. The tool may otherwise move on a wrong path.

To program a helix:

▶ Select **C**

▶ Select **P**

▶ 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
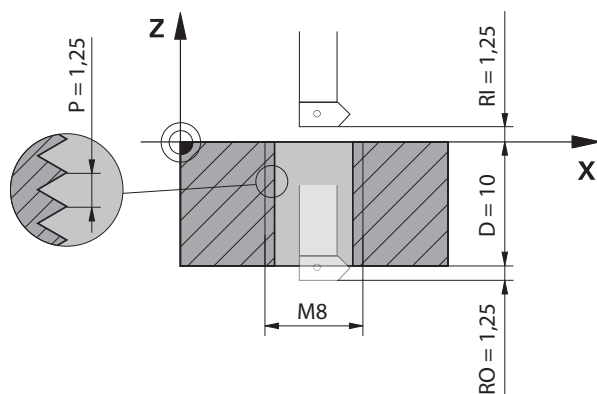
This example includes the following default values:

- **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 328

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

Alternatively, you can also program the thread with a program section repeat.

Further information: "Subprograms and program section repeats with the label LBL", Page 360

Further information: "Example", Page 316

11.4.7 Example: Polar straight lines
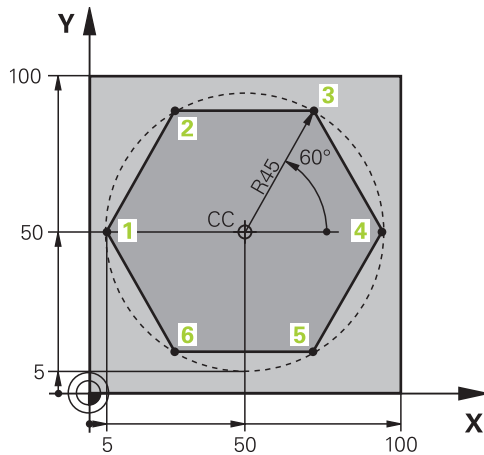
0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Workpiece blank definition
3 TOOL CALL 1 Z S4000	; Tool call
4 CC X+50 Y+50	; Define the datum for polar coordinates
5 L Z+250 R0 FMAX	; Retract the tool
6 LP PR+60 PA+180 R0 FMAX	; Pre-position the tool
7 L Z-5 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 Fundamentals of approach and departure functions

Approach and departure functions allow you to avoid dwell marks on the workpiece because the tool gently approaches and departs from the contour.

Because the approach and departure functions encompass multiple path functions, you get shorter NC programs. The defined syntax elements **APPR** and **DEP** make it easier for you to find contours in the NC program.

11.5.1 Overview of the approach and departure functions

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 333
	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 336
	APPR CT or **APPR PCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection	Page 338
	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 340

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 342
	DEP LN Depart contour on a straight line perpendicular to the last contour point	Page 343
	DEP CT Depart contour on a circular path with a tangential connection	Page 344
	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 344

> **i** 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 292

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: "Linear superpositioning of a circular path", Page 327

11.5.2 Positions for approach and departure
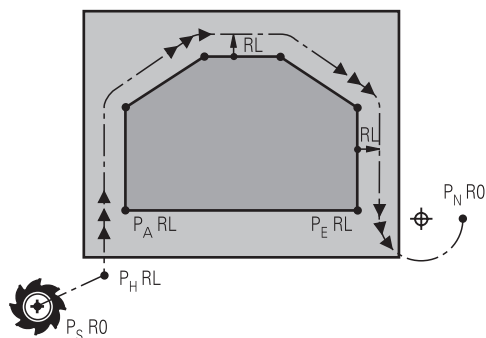
> **NOTICE**
>
> **Danger of collision!**
>
> The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **FMAX** in the last positioning block before the approach function, the control also approaches the auxiliary 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

The starting point P_S is programmed prior to the approach function without radius compensation. The starting point is located outside of the contour.

■ Auxiliary point P_H

Certain approach and departure functions require an additional auxiliary point P_H. The control automatically calculates the auxiliary point using the entered information.

In order to determine the auxiliary point P_H, the control requires a subsequent path function. If no path function follows, then the control stops the machining operation or simulation with an error message.

■ First contour point P_A

Program the first contour point P_A within the approach function, along with the radius compensation **RR** or **RL**.

> **i**   If you program **R0**, then the control may stop the machining operation or simulation with an error message.
>
> This reaction is different from the behavior of the iTNC 530.

■ 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 arises from the information entered within the departure function. The departure function automatically cancels 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

---

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

11.6 Approach and departure functions with Cartesian coordinates

11.6.1 Approach function APPR LT

Application

With the **APPR LT** NC function, the control approaches the contour on a straight line tangential to the first contour element.

Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

■ **APPR PLT** with polar coordinates
   Further information: "Approach function APPR PLT", Page 347

Description of function
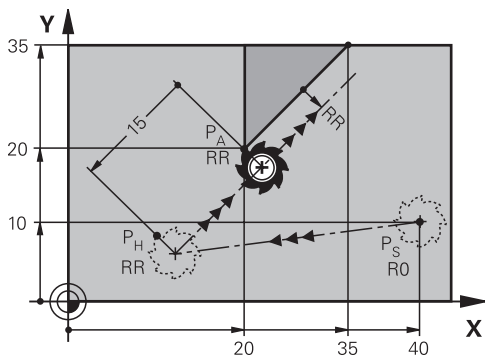
This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
- A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR LT X+20 Y+20 LEN15 RR F300	; Approach the contour on a tangential linear path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR LT

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LT	Syntax initiator for a linear approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR LT

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	; Approach P_A with **RR**, distance P_H to P_A: **LEN15**
13 L X+35 Y+35	; Complete the first contour element

11.6.2   Approach function APPR LN

Application

With the NC function **APPR LN**, the control approaches the contour on a straight line perpendicular to the first contour element.

Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

■ **APPR PLN** with polar coordinates
  Further information: "Approach function APPR PLN", Page 349

Description of function
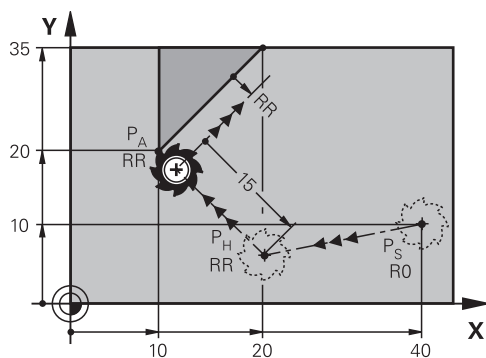
This NC function encompasses the following steps:

■ A straight line from the starting point P_S to the auxiliary point P_H

■ A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR LN X+20 Y+20 LEN+15 RR F300	; Linearly and perpendicularly approach the contour

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR LN

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LN	Syntax initiator for a linear approach function perpendicular to the contour
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.
Further information: "The Form column in the Program workspace", Page 214

Example APPR LN

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LN X+10 Y+20 Z-10 LEN+15 RR F100	; Approach P_A with **RR**; distance: P_H to P_A: **LEN+15**
13 L X+20 Y+35	; Complete the first contour element

11.6.3 Approach function APPR CT

Application

With the NC function **APPR CT**, the control approaches the contour on a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PCT** with polar coordinates
  Further information: "Approach function APPR PCT", Page 351

Description of function

This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H

  The distance of the auxiliary point P_H to the first contour point P_A arises from the center angle **CCA** and the radius **R**.

- A circular path from the auxiliary point P_H to the first contour point P_A

  The circular path is defined by the center angle **CCA** and the radius **R**.

  The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> **ℹ** If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 APPR CT X+20 Y+20 CCA80 R+5 RR F300	; Approach the contour on a tangential circular path

To navigate to this function:

Insert NC function ▸ All functions ▸ Path contour ▸ APPR ▸ APPR CT

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR CT	Syntax initiator for a circular approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
CCA	Center angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR CT

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR CT X+10 Y+20 Z-10 CCA180 R +10 RR F100	; Approach P_A with **CCA180** and **RR**; distance P_H to P_A: **R+10**
13 L X+20 Y+35	; Complete the first contour element

11.6.4 Approach function APPR LCT

Application

With the NC function **APPR LCT**, the control approaches the contour on a straight line, followed by a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

■ **APPR PLCT** with polar coordinates

Further information: "Approach function APPR PLCT", Page 354

Description of function

This NC function encompasses the following steps:

■ A straight line from the starting point P_S to the auxiliary point P_H

The straight line is tangential to the circular path.

The auxiliary point P_H is determined based on the starting point P_S, the radius **R** and the first contour point P_A.

■ A circular path in the working plane from the auxiliary point P_H to the first contour point P_A

The circular path is uniquely defined by the radius **R**.

If you program the Z coordinates in the approach function, then the tool approaches simultaneously in three axes from the starting point P_S to the auxiliary point P_H.

Input

11 APPR LCT X+20 Y+20 Z-10 R5 RR F300	; Approach the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR LCT

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR LCT	Syntax initiator for a linear and circular approach function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C, U**, **V**, **W**	Coordinates of the first contour point Fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR LCT

11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LCT X+10 Y+20 Z-10 R10 RR F100	; Approach P_A with **RR**; distance P_H to P_A: **R10**
13 L X+20 Y+35	; Complete the first contour element

11.6.5 Departure function DEP LT

Application

With the NC function **DEP LT**, the control departs from the contour on a straight line tangential to the last contour element.

Description of function
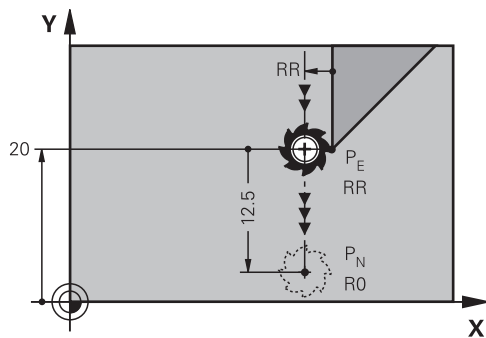
The tool moves in a straight line from the last contour point P_E to the end point P_N.

Input

11 DEP LT LEN5 F300	; Depart from the contour on a tangential linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LT	Syntax initiator for a linear departure function tangential to the contour
LEN	Distance of the auxiliary point P_H to the contour
	Fixed or variable number
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Example DEP LT

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LT LEN12.5 F100	; Approach P_N; distance P_E to P_N: **LEN12.5**

11.6.6 Departure function DEP LN

Application

With the NC function **DEP LN**, the control departs from the contour on a straight line perpendicular to the last contour element.

Description of function
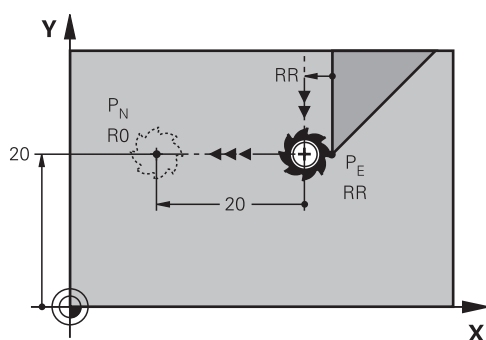
The tool moves in a straight line from the last contour point P_E to the end point P_N.

The distance from the end point P_N to the contour point P_E is **LEN** plus the tool radius.

Input

11 DEP LN LEN+10 F300	; Depart from the contour on a perpendicular linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LN**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LN	Syntax initiator for a linear departure function perpendicular to the contour
LEN	Distance of the auxiliary point P_H to the contour
	Fixed or variable number
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Example DEP LN

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LN LEN+20 F100	; Approach P_N; distance P_E to P_N: **LEN+20**

11.6.7   Departure function DEP CT

Application

With the NC function **DEP CT**, the control departs from the contour on a circular path tangential to the last contour element.

Description of function
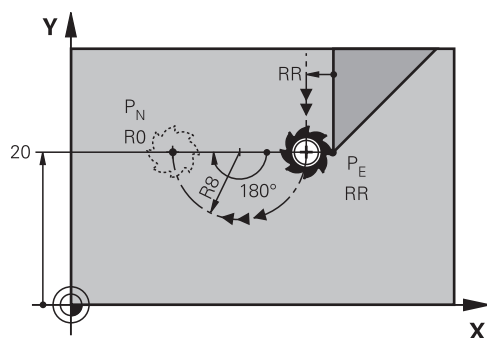
The tool moves on a circular path from the last contour point P_E to the end point P_N.

The circular path is defined by the center angle **CCA** and the radius **R**.

The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> **i**   If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 DEP CT CCA30 R+8	; Depart from the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP CT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP CT	Syntax initiator for a circular departure function tangential to the contour
CCA	Center angle as a fixed or variable number
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Example DEP CT

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP CT CCA180 R+8 F100	; Approach P_N with **CCA180**; distance P_E to P_N: **R+8**

11.6.8 Departure function DEP LCT

Application

With the NC function **DEP LCT**, the control departs from the contour on a circular path, followed by a tangential straight line to the last contour element.

The coordinates of the end point P_N are programmed with Cartesian coordinates.

Related topics

■ **DEP LCT** with polar coordinates
   Further information: "Departure function DEP PLCT", Page 356

Description of function

This NC function encompasses the following steps:

- On a circular path from the last contour point P_E to the auxiliary point P_H

  The auxiliary point P_H is determined based on the last contour point P_E, the radius **R** and the end point P_N.

- 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 function, then the tool moves simultaneously in three axes from the auxiliary point P_H to the end point P_N.

Input

11 DEP LCT X-10 Y-0 R15	; Tangentially depart from the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **DEP** ▶ **DEP LCT**

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP LCT	Syntax initiator for a linear and circular departure function tangential to the contour
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V**, **W**	Coordinates of the last contour point Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example DEP LCT

11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LCT X+10 Y+12 R8 F100	; Approach P_N; distance P_E to P_N: **R8**

11.7 Approach and departure functions with polar coordinates

11.7.1 Approach function APPR PLT

Application

With the **APPR PLT** NC function, the control approaches the contour on a straight line tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

Related topics

■ **APPR LT** with Cartesian coordinates

Further information: "Approach function APPR LT", Page 333

Requirement

■ Pole **CC**

You must define a pole **CC** before programming with polar coordinates.

Further information: "Polar coordinate datum at pole CC", Page 319

Description of function
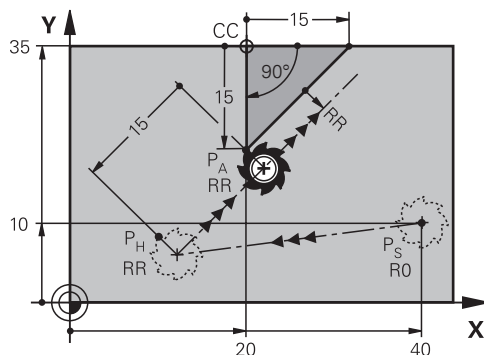
This NC function encompasses the following steps:

■ A straight line from the starting point P_S to the auxiliary point P_H

■ A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR PLT PR+15 PA-90 LEN15 RR F200	; Approach the contour on a tangential linear path

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Path contour** ▶ **APPR** ▶ **APPR PLT**

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLT	Syntax initiator for a linear approach function tangential to the contour
PR	Polar coordinate radius
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Entry: absolute or incremental
	Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour
	Fixed or variable number
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 798
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR PLT

11 L X+10 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLT PR+30 PA+180 LEN10 RL F300	; Approach P_A with **RL**; distance from P_H to P_A: **LEN10**
14 LP PR+30 PA+125	; Complete the first contour element

11.7.2 Approach function APPR PLN

Application

With the NC function **APPR PLN**, the control approaches the contour on a straight line perpendicular to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

Related topics

■ **APPR LN** with Cartesian coordinates
  Further information: "Approach function APPR LN", Page 336

Requirement

■ Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 319

Description of function

This NC function encompasses the following steps:

■ A straight line from the starting point P_S to the auxiliary point P_H
■ A straight line from the auxiliary point P_H to the first contour point P_A

Input

11 APPR PLN PR+15 PA-90 LEN+15 RL F300	; Linearly and perpendicularly approach the contour

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR PLN

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLN	Syntax initiator for a linear approach function perpendicular to the contour
PR	Polar coordinate radius Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Entry: absolute or incremental Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR PLN

11 L X-5 Y+25 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLN PR+30 PA+180 LEN+10 RL F300	; Approach P_A with **RL**; P_H to P_A; **LEN+10**
14 LP PR+30 PA+125	; Complete the first contour element

11.7.3 Approach function APPR PCT

Application

With the NC function **APPR PCT**, the control approaches the contour on a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

Related topics

■ **APPR CT** with Cartesian coordinates
Further information: "Approach function APPR CT", Page 338

Requirement

■ Pole **CC**
You must define a pole **CC** before programming with polar coordinates.
Further information: "Polar coordinate datum at pole CC", Page 319

Description of function

This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H

  The distance of the auxiliary point P_H to the first contour point P_A arises from the center angle **CCA** and the radius **R**.

- A circular path from the auxiliary point P_H to the first contour point P_A

  The circular path is defined by the center angle **CCA** and the radius **R**.

  The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> **i** If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 APPR PCT PR+15 PA-90 CCA180 R +10 RL F300	; Approach the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR PCT

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PCT	Syntax initiator for a circular approach function tangential to the contour
PR	Polar coordinate radius Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Entry: absolute or incremental Optional syntax element
CCA	Center angle as a fixed or variable number Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR PCT

11 L X+5 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PCT PR+30 PA+180 CCA40 R +20 RL F300	; Approach P_A with **CCA40** and **RL**; distance P_H to P_A: **R+20**
14 LP PR+30 PA+125	; Complete the first contour element

11.7.4 Approach function APPR PLCT

Application

With the NC function **APPR PLCT**, the control approaches the contour on a straight line, followed by a circular path tangential to the first contour element.

Coordinates of the first contour point are programmed with polar coordinates.

Related topics

- **APPR LCT** with Cartesian coordinates
  Further information: "Approach function APPR LCT", Page 340

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 319

Description of function

This NC function encompasses the following steps:

- A straight line from the starting point P_S to the auxiliary point P_H
  The straight line is tangential to the circular path.
  The auxiliary point P_H is determined based on the starting point P_S, the radius **R** and the first contour point P_A.
- A circular path in the working plane from the auxiliary point P_H to the first contour point P_A
  The circular path is uniquely defined by the radius **R**.

If you program the Z coordinates in the approach function, then the tool approaches simultaneously in three axes from the starting point P_S to the auxiliary point P_H.

Input

11 APPR PLCT PR+15 PA-90 R10 RL F300	; Tangentially approach the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ APPR ▶ APPR PLCT

The NC function includes the following syntax elements:

Syntax element	Meaning
APPR PLCT	Syntax initiator for a linear and circular approach function tangential to the contour
PR	Polar coordinate radius Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Entry: absolute or incremental Optional syntax element
R	Radius as a fixed or variable number Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 798 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 284 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 963 Fixed or variable number Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example APPR PLCT

11 L X+10 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 CC X+50 Y+20	; Set the pole
13 APPR PLCT PR+30 PA+180 R20 RL F300	; Approach P_A with **RL**; P_H to P_A: **R20**
14 LP PR+30 PA+125	; Complete the first contour element

11.7.5   Departure function DEP PLCT

Application

With the NC function **DEP PLCT**, the control departs from the contour on a circular path, followed by a tangential straight line to the last contour element.

The coordinates of the end point P_N are programmed with polar coordinates.

Related topics

- **DEP LCT** with Cartesian coordinates
  Further information: "Departure function DEP LCT", Page 345

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 319

Description of function

This NC function encompasses the following steps:

- On a circular path from the last contour point P_E to the auxiliary point P_H
  The auxiliary point P_H is determined based on the last contour point P_E, the radius **R** and the end point P_N.
- 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 function, then the tool moves simultaneously in three axes from the auxiliary point P_H to the end point P_N.

Input

11 DEP PLCT PR15 PA-90 R8	; Tangentially depart from the contour linearly and circularly

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ DEP ▶ DEP PLCT

The NC function includes the following syntax elements:

Syntax element	Meaning
DEP PLCT	Syntax initiator for a linear and circular departure function tangential to the contour
PR	Polar coordinate radius
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Entry: absolute or incremental
	Optional syntax element
R	Radius as a fixed or variable number
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 284
	Fixed or variable number
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 963
	Fixed or variable number
	Optional syntax element

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate input.

Further information: "The Form column in the Program workspace", Page 214

Example DEP PLCT

11 CC X+50 Y+20	; Set the pole
12 LP PR+30 PA+0 RL F300	; Approach the last contour element P_E with **RL**
13 DEP PLCT PR+50 PA+0 R5	; Approach P_N; distance P_E to P_N: **R5**

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: "Call the NC program with CALL PGM", Page 364
- Jumps with conditions as if-then decisions.

  Further information: "The Jump commands folder", Page 1026

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
↻ ¹₂...	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 its number or name. If a number or a name exists twice in an NC program, the control shows a warning before the NC block.

LBL 0 marks the end of a subprogram. This number is the only one which may exist more than once in the NC program.

Input

11 LBL "Reset"	; Subprogram for resetting a coordinate transformation
12 TRANS DATUM RESET	
13 LBL 0	

To navigate to this function:

Insert NC function ▶ All functions ▶ Label ▶ LBL SET

The NC function includes the following syntax elements:

Syntax element	Meaning
LBL	Syntax initiator for a label
Number or **Name**	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 360

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Label ▶ CALL LBL

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL LBL	Syntax initiator for calling a label
Number, **Name**, 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 that displays all labels available in the NC program.
REP	Number of repetitions until the control executes the next NC block
	Optional syntax element

Subprograms

```
0   BEGIN PGM ...
        ①
    CALL LBL1
        ②
    L Z+100 M2
    LBL1
        ③
    LBL0
    END PGM ...
```
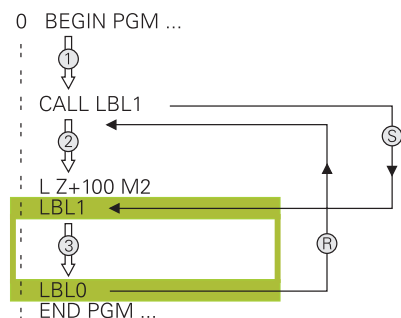
A subprogram allows calling parts of an NC program any number of times at different points of the NC program (e.g., machining positions or a contour).

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 161

Program-section repeats

0   BEGIN PGM ...

LBL1

CALL LBL 1 REP 2

END PGM ...
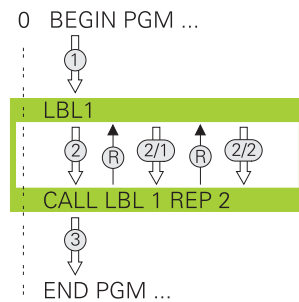
A program section repeat allows repeating a part of an NC program any number of times (e.g., contour machining with incremental infeed).

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 161

Notes

■   The control displays the NC function **LBL SET** in the structure by default.

Further information: "The Structure column in the Program workspace", Page 1166

■   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> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

12.2   Selection functions

12.2.1   Overview of selection functions

The **Selection** folder of the **Insert NC function** window contains the following functions:

Icon	Meaning	Further information
CALL PGM	Call an NC program with **CALL PGM**	Page 364
⊕	Select a datum table with **SEL TABLE**	Page 716
░░░	Select a point table with **SEL PATTERN**	Page 393
⬠	Select a contour program with **SEL CONTOUR**	Page 386
📄	Select an NC program with **SEL PGM**	Page 366
📄	Call the last selected file with **CALL SELECTED PGM**	Page 366
CYC	Select any NC program with **SEL CYCLE** as a machining cycle	Page 227
▦	Select a correction table with **SEL CORR-TABLE**	Page 801
🗋	Open the file with **OPEN FILE**	Page 843
	Link multiple contours with **CONTOUR DEF**	Page 380

12.2.2   Call the NC program with CALL PGM

Application

With the **CALL PGM** NC function, you can call another, separate NC program from within an NC program. The control executes the called NC program at the point where you called it in the NC program. This allows a machining operation to be executed with various transformations, for example.

Related topics

■ Program call with Cycle **12 PGM CALL**

 Further information: "Cycle 12 PGM CALL ", Page 368

■ Program call following selection

 Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 366

■ Executing multiple NC programs as a job list

 Further information: "Pallet Machining and Job Lists", Page 1619

Description of function
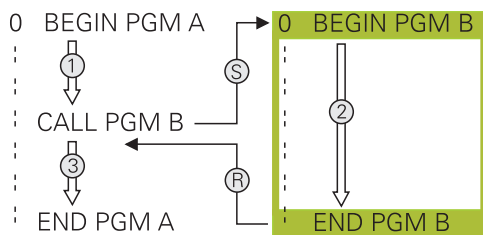
The control executes the NC program as follows:

1 The control executes the calling NC program until you call another NC program with **CALL PGM**.

2 After that, the control executes the called NC program up to the last NC block.

3 The control then resumes the calling NC program, starting with 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 the miscellaneous function **M30** or **M2**. If you defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

Further information: "Unconditional jump", Page 1027

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Selection ▶ CALL PGM

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL PGM	Syntax initiator for calling an NC program
File	Path of the called NC program Selection by means of a selection window

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 conjunction with string parameters, use the **SEL PGM** NC function.
   Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 366

■ As a rule, Q parameters are globally effective when used with a program call, such as **CALL PGM**. So please note that changes made to Q parameters in the called NC program also influence the calling NC program. If applicable, use QL parameters that take effect 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: "Call the NC program with CALL PGM", Page 364

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 the miscellaneous function **M30** or **M2**. If you defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

  Further information: "Unconditional jump", Page 1027

  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

SEL PGM

To navigate to this function:

Insert NC function ▶ All functions ▶ Selection ▶ SEL PGM

The NC function includes the following syntax elements:

Syntax element	Meaning
SEL PGM	Syntax initiator for selecting an NC program to be called
Name or **QS**	Path of the NC program to be called
	Fixed or variable path
	Selection by means of a selection window

CALL SELECTED PGM

To navigate to this function:

Insert NC function ▶ All functions ▶ Selection ▶ CALL SELECTED PGM

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL SELECTED PGM	Syntax for calling the selected NC program

Notes

■ Within the **SEL PGM** NC 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 the program run, you can use the **FN 18: SYSREAD** (**ID10 NR110** and **NR111**) NC function to check all paths at program start.

Further information: "Read system data with FN 18: SYSREAD", Page 1035

■ 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 globally effective when used with a program call, such as **CALL PGM**. So please note that changes made to Q parameters in the called NC program also influence the calling NC program. If applicable, use QL parameters that take effect 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 12 PGM CALL

ISO programming

G39

Application
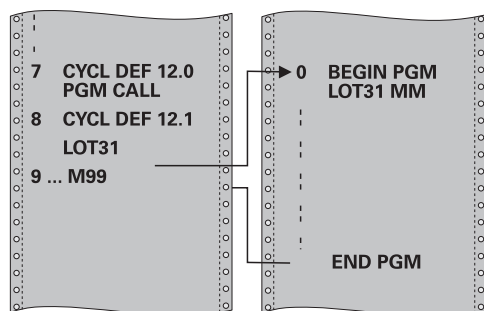
NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.

Related topics

■ Calling external NC programs

Further information: "Selection functions", Page 364

Notes

■ This cycle can be executed in the **FUNCTION MODE MILL** machining mode.

■ 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**.
- If you want to define an ISO program to be a cycle, add the .I file type to the program name.

12.3.1 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

12.4 NC sequences for reuse

Application

You can save up to 200 consecutive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. Unlike called NC programs, you can modify NC sequences after insertion without changing the actual sequence.

Related topics

- **Insert NC function** window
  Further information: "Areas of the Insert NC function window", Page 216
- Mark and copy NC blocks with the context menu
  Further information: "Context menu", Page 1174
- Call NC programs unchanged
  Further information: "Call the NC program with CALL PGM", Page 364

Description of function

You can use NC sequences in the **Editor** operating mode and the **MDI** application.

The control saves the NC sequences as complete NC programs in the **TNC:\system\PGM-Templates** folder. You can also create subfolders in order to sort the NC sequences.

Here are the following possibilities for creating an NC sequence:

- Save marked NC blocks with the **Create NC sequence** button

  Further information: "Context menu in the Program workspace", Page 1177

- Create a new NC program in the **TNC:\system\PGM-Templates** folder
- Copy the already existing NC program to the **TNC:\system\PGM-Templates** folder

If you create an NC sequence with the **Create NC sequence** button, then the control opens the **Save NC sequence** window.

In the **Save NC sequence** window, you can enter the following information:

- Define the name of the NC sequence
- Select the storage location of the NC sequence

  If you created subfolders in the **TNC:\system\PGM-Templates** folder, the control will display a selection menu that contains all folders.

The control displays all folders and NC sequences alphabetically in the **Insert NC function** window under **NC sequences**. You can insert the desired NC sequence at the cursor position and customize it in the NC program.
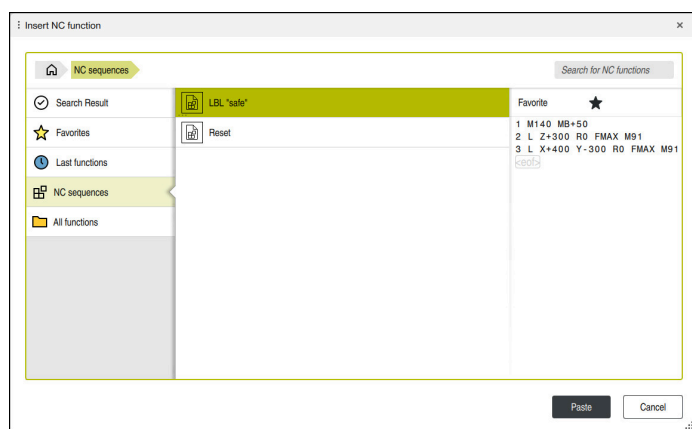
Inserting NC sequences in the **Insert NC function** window

If you open an NC sequence as its own tab in the **Editor**, then you can permanently edit the contents of the NC sequence.

Notes

- Make sure to define an unambiguous name for each NC sequence within a folder. If you try to save an NC sequence under a name that has already been assigned, then the control opens the **Overwrite NC sequence** window. The control asks if you wish to overwrite the existing NC sequence.
- If you drag an NC sequence to the right in the **Insert NC function** window, the control will display the following file functions:
  - Edit
  - Rename
  - Delete
  - Activate or deactivate write protection
  - Open the path in the **Files** operating mode
  - Mark as favorite

  Further information: "Context menu in the Insert NC function window", Page 1178
- Write-protected NC sequences cannot be renamed or deleted. It is possible to edit such an NC sequence, but you need to save it as a new file after editing.

  While write protection is active, the control displays a corresponding symbol next to the NC sequence.
- If you create a backup of the **TNC:** partition with the **NC/PLC Backup** function, then the backup also contains the NC sequences.

  Further information: "Backup and restore", Page 1829
- If you insert an NC sequence into an NC program, the control will not convert the mm and inch units of measure. Ensure that the unit of measure used in the NC sequence matches the one used in the NC program.

12.5 Nesting of programming techniques

Application

It is possible to combine programming techniques, for example when calling a separate NC program or subprogram from within a program-section repeat.

If you want to return to the origin after each call, use only one nesting level. If you program another call before returning to the origin, you will get one nesting level lower.

Related topics

- Subprograms

  Further information: "Subprograms", Page 362
- Program section repeats

  Further information: "Program-section repeats", Page 363
- Calling a separate NC program

  Further information: "Selection functions", Page 364

Description of function

Please note the maximum nesting depth:

- Maximum nesting depth for subprogram calls: 19
- Maximum nesting depth for calls of external NC programs: 19 where a **CYCL CALL** has the same effect as calling an external program
- Program-section repeats can be nested 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 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 to NC block 0.

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 0.

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 0.

13

Contour and Point
Definitions

13.1 Superimposing contours

13.1.1 Fundamentals
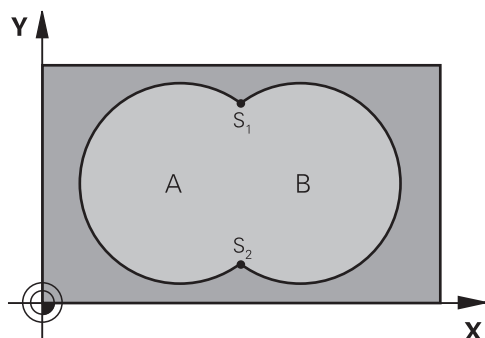
Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Related topics

- Cycle 14 **CONTOUR**

    Further information: "Cycle 14 CONTOUR ", Page 379

- SL cycles

    Further information: "Milling contours with SL cycles ", Page 586

- OCM cycles

    Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626

13.1.2 Subprograms: overlapping pockets

> **i** 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

13.1.3 Surface resulting from sum
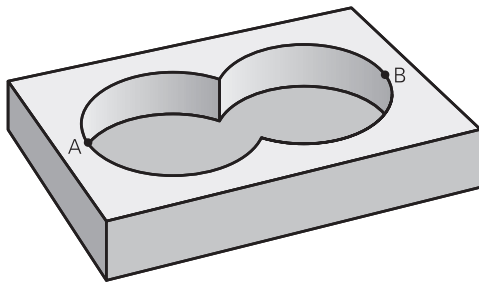
Both surfaces A and B are to be machined, including the overlapping area:

- 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

13.1.4 Surface resulting from difference
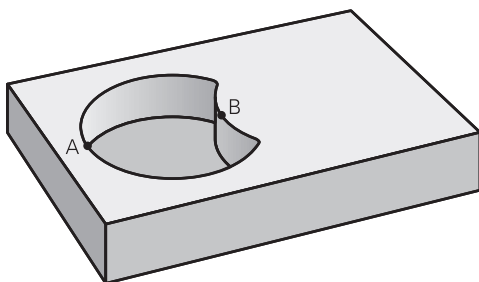
Surface A is to be machined without the portion overlapped by B:

- 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

13.1.5 Surface resulting from intersection
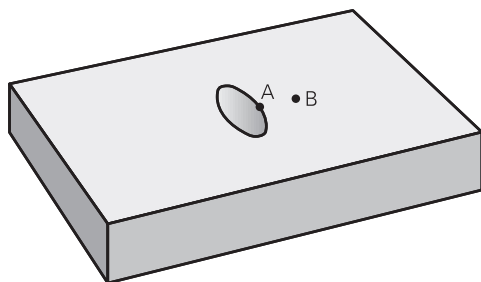
Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- 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

13.2 Cycle 14 CONTOUR

ISO programming

G37

Application
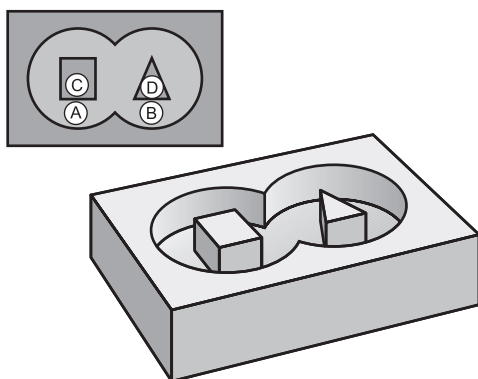
In Cycle **14 CONTOUR**, list all subprograms that are to be superimposed to define the overall contour.

Related topics

- Simple contour formula

  Further information: "Simple contour formula", Page 380

- Complex contour formula

  Further information: "Complex contour formula", Page 383

- Superimposing contours

  Further information: "Superimposing contours", Page 376

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 takes effect as soon as it is defined in the NC program.

- You can list up to 12 subprograms (subcontours) in Cycle **14**.

13.2.1 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

13.3 Simple contour formula

13.3.1 Fundamentals

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.

Related topics

■ Superimposing contours

Further information: "Superimposing contours", Page 376

■ Complex contour formula

Further information: "Complex contour formula", Page 383

■ Cycle 14 **CONTOUR**

Further information: "Cycle 14 CONTOUR ", Page 379

■ SL cycles

Further information: "Milling contours with SL cycles ", Page 586

■ OCM cycles

Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626

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

> **ⓘ** The memory capacity for programming an SL cycle (all contour description programs) is limited to **100 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 areas 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 set-up clearance, 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

> **i** 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
```

> **i** 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 in effect 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

> ℹ️ The entire contour is machined with the SL cycles (see "Milling contours with SL cycles ", Page 586) or the OCM cycles (see "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626).

13.4 Complex contour formula

13.4.1 Fundamentals
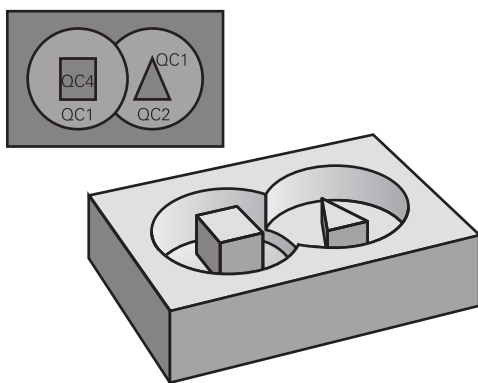
Using complex contour formulas, you can combine several subcontours (pockets or islands) to program complex contours. You define the individual subcontours (geometry data) in separate NC programs or subprograms. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours, which you link through a contour formula.

Related topics

- Superimposing contours

  Further information: "Superimposing contours", Page 376

- Simple contour formula

  Further information: "Simple contour formula", Page 380

- Cycle 14 **CONTOUR**

  Further information: "Cycle 14 CONTOUR ", Page 379

- SL cycles

  Further information: "Milling contours with SL cycles ", Page 586

- OCM cycles

  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626

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

> **i** Programming notes:
> - The memory capacity for programming an SL cycle (all contour description programs) is limited to **100 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:

Insert NC function	▶	Select **Insert NC function**
	>	The control opens the **Insert NC function** window.
[icon]	▶	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
LBL	▪ Number ▪ Name ▪ QS	Define the number, name or QS parameter for a label

i 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:

Insert
NC function

- ▶ 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		Specify the number of a string parameter
LBL	▪ Number ▪ Name ▪ QS	Define the number, name or QS parameter for a label

ⓘ 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.
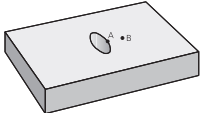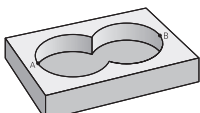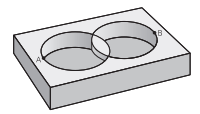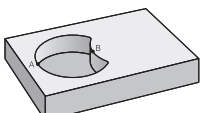
  Further information: "Simple contour formula", Page 380

13.4.4 Entering a complex contour formula

You can use the contour formula function to interlink various contours in a mathematical formula.

Insert NC function

▶ 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

Help graphic	Input	Mathematical function	Example
	&	Intersected with	**QC10 = QC1 & QC2**
	|	Joined with	**QC10 = QC1 | QC2**
	^	Joined with, but w/o intersection	**QC10 = QC1 ^ QC2**
	\	Without	**QC10 = QC1 \ QC2**
	(	Opening parenthesis	**QC10 = QC1 & (QC2 | QC3)**
	)	Closing parenthesis	**QC10 = QC1 & (QC2 | QC3)**
		Defining a single contour	**QC10 = QC1**

The control provides the following options to enter formulas:

■ Auto-complete
  Further information: "Entering a formula using the auto-complete function", Page 1047
■ Pop-up keyboard for formula input from the action bar or from within the form
■ Formula input mode of the virtual keyboard
  Further information: "Virtual keyboard of the control bar", Page 1160

13.4.5 Superimposed contours
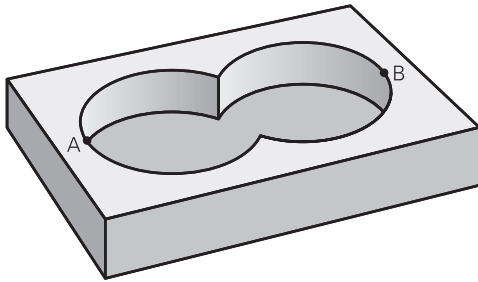
By default, the control considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

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

> ℹ 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
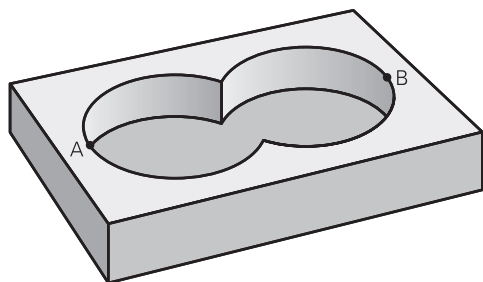
Both areas A and B are to be machined, including the overlapping area:

- 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
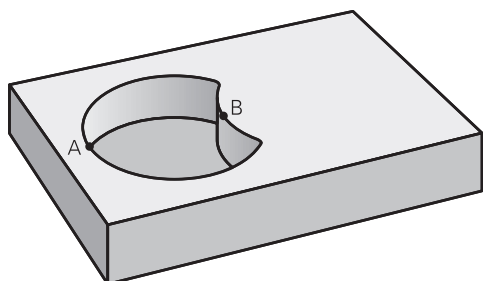
Area A is to be machined without the portion overlapped by B:

- 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 using 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
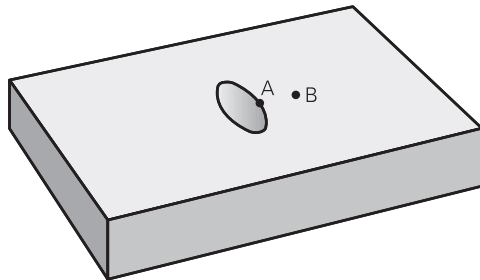
Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- 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

> **i** The entire contour is machined with the SL cycles (see "Milling contours with SL cycles ", Page 586) or the OCM cycles (see "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626).

13.5 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 *.pnt", Page 1720

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.

> ℹ️ 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.

13.5.1 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 206

Example

7 SEL PATTERN "TNC:\nc_prog\Positions.PNT

13.5.2 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**.

13.6 Pattern definition with PATTERN DEF

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.

Related topics

- Cycles for pattern definition
  Further information: "Pattern definition cycles", Page 405

> *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**

To navigate to this function:

Insert NC function ▶ Contour/point machining ▶ Pattern

Possible setting	Definition	Further information
POS	Point Definition of up to any 9 machining positions	Page 396
ROW	Row Definition of a single row, straight or rotated	Page 397
PAT	Pattern Definition of a single pattern, straight, rotated or distorted	Page 398
FRAME	Frame Definition of a single frame, straight, rotated or distorted	Page 400
CIRC	Circle Definition of a full circle	Page 401
PITCH-CIRC	Pitch circle Definition of a pitch circle	Page 402

Programming PATTERN DEF

To program the **PATTERN DEF** functions:

Insert NC function	▶ Select **Insert NC function**
	> The control opens the **Insert NC function** window.
	▶ Select the desired machining pattern (e.g., **PATTERN DEF CIRC** for a full circle)
	> The control opens the dialog for entering **PATTERN DEF**.
	▶ Enter the required definitions
	▶ Define the machining cycle (e.g., Cycle **200**) **DRILLING**
	▶ Call cycle with **CYCL CALL PAT**

> **i** While you are programming a machining pattern, you can switch to a different machining pattern in the **Form** column.

Calling PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function.

Further information: "Calling cycles", Page 227

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 393

■ 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 1649

13.6.1 Defining individual machining positions

> **i**   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)

13.6.2 Defining a single row

> **i** 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. This 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. This 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)

13.6.3 Defining an individual pattern

> **i** 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**
	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**

Help graphic	Parameter
	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)

13.6.4 Defining an individual frame

ℹ️ 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**
	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**

Help graphic	Parameter
	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)

13.6.5 Defining a full circle

> **i** 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)

13.6.6 Defining a pitch circle

i 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)

13.6.7 Example: Using cycles in conjunction 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: "Cycles for Drilling, Centering and Thread Machining", Page 445 and "Milling Cycles"

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	
20 M30	; End of program	
21 END PGM 1 MM		

13.7 Pattern definition cycles

13.7.1 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 407
221	**CARTESIAN PATTERN** ■ Defining a linear pattern ■ Input of an angle of rotation	**DEF**-active	Page 410
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 414

You can combine the following cycles with point pattern cycles:

	Cycle 220	Cycle 221	Cycle 224
200 DRILLING	✓	✓	✓
201 REAMING	✓	✓	✓
202 BORING	✓	✓	–
203 UNIVERSAL DRILLING	✓	✓	✓
204 BACK BORING	✓	✓	–
205 UNIVERSAL PECKING	✓	✓	✓
206 TAPPING	✓	✓	–
207 RIGID TAPPING	✓	✓	–
208 BORE MILLING	✓	✓	✓
209 TAPPING W/ CHIP BRKG	✓	✓	–
240 CENTERING	✓	✓	✓
251 RECTANGULAR POCKET	✓	✓	✓
252 CIRCULAR POCKET	✓	✓	✓
253 SLOT MILLING	✓	✓	–
254 CIRCULAR SLOT	–	✓	–
256 RECTANGULAR STUD	✓	✓	–
257 CIRCULAR STUD	✓	✓	–
262 THREAD MILLING	✓	✓	–
263 THREAD MLLNG/ CNTSNKG	✓	✓	–
264 THREAD DRILLNG/MLLNG	✓	✓	–
265 HEL. THREAD DRLG/MLG	✓	✓	–
267 OUTSIDE THREAD MLLNG	✓	✓	–

> **ℹ** If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.
> More regular point patterns are available with the **PATTERN DEF** function.

Further information: "Point tables", Page 391
Further information: "Pattern definition with PATTERN DEF", Page 394

13.7.2 Cycle 220 POLAR PATTERN

ISO programming

G220

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.

Related topics

- Defining a full circle with **PATTERN DEF**

    Further information: "Defining a full circle", Page 401

- Defining a circle segment with **PATTERN DEF**

    Further information: "Defining a pitch circle", Page 402

Cycle run

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

> ⚙ Cycle **220 POLAR PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

- 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
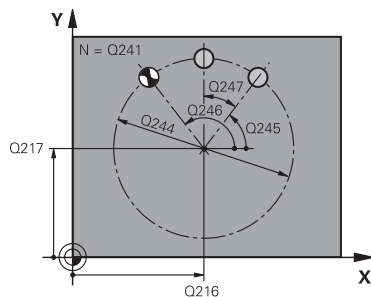
Help graphic	Parameter
	Q216 Center in 1st axis? Pitch circle center in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis? Pitch circle center in the secondary axis of the working plane. This value has an absolute effect. 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. This 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. This 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. This 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	

13.7.3 Cycle 221 CARTESIAN PATTERN

ISO programming

G221

Application
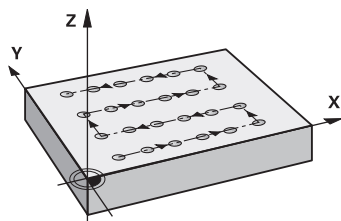
This cycle enables you to define a point pattern as lines. It can be used for a previously defined machining cycle.

Related topics

- Defining an individual row with **PATTERN DEF**

  Further information: "Defining a single row", Page 397

- Defining an individual pattern with **PATTERN DEF**

  Further information: "Defining an individual pattern", Page 398

Cycle run

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 row

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

> ⚙ Cycle **221 CARTESIAN PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming
- If you combine Cycle **221** with one of the machining cycles **200** to **209** or **251** to **267**, then the set-up clearance, the workpiece surface, the 2nd set-up clearance, and the rotary position that you defined in Cycle **221** will be effective for the selected machining cycle.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

Cycle parameters
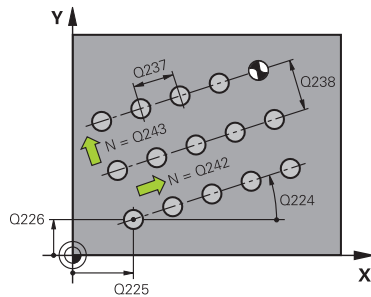
Help graphic	Parameter
	Q225 Starting point in 1st axis? Coordinate of starting point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate of starting point in the secondary axis of the working plane. This 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. This 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. This 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

Help graphic	Parameter
	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		

13.7.4 Cycle 224 DATAMATRIX CODE PATTERN

ISO programming

G224

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
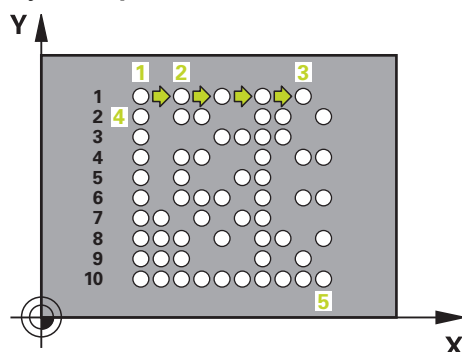
1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.

Sequence:

- Move to 2nd set-up clearance (spindle axis)
- Approach the starting point in the working plane
- Move to **SET-UP 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. There is a danger of collision! ▶ Check the machining sequence using a graphic simulation ▶ Carefully test the NC program or program section in **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
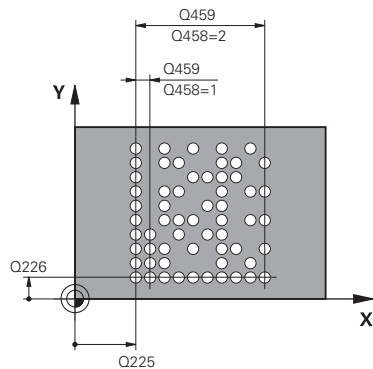
Help graphic	Parameter
	Q225 Starting point in 1st axis? Coordinate in the lower left corner of the code in the main axis. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate in the bottom left corner of the data matrix code in the secondary axis. This 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 417 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. This 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. This 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.

Help graphic	Parameter
	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.)

> **i**   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 counter reading into a DataMatrix code. The current counter reading is displayed during **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

- During simulation, the control only simulates the counter reading that you define directly in the NC program. The counter reading from the **Status** workspace of the **Program Run** operating mode is ignored.

13.7.5 Programming examples

Example: Polar hole patterns
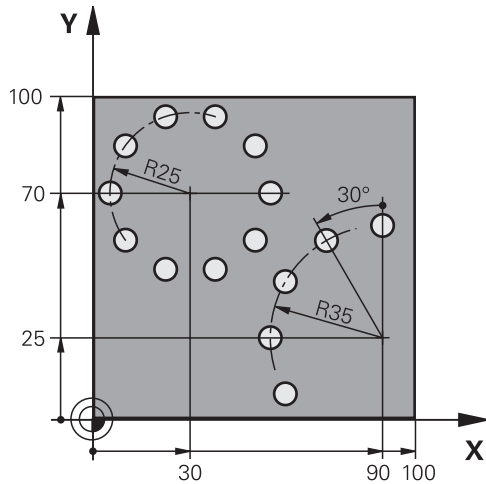
0	BEGIN PGM 200 MM	
1	**BLK FORM 0.1 Z X+0 Y+0 Z-40**	
2	**BLK FORM 0.2 X+100 Y+100 Z+0**	
3	**TOOL CALL 200 Z S3500**	; Tool call
4	**L Z+100 R0 FMAX M3**	; Retract the tool
5	**CYCL DEF 200 DRILLING ~**	
	Q200=+2 ;SET-UP CLEARANCE ~	
	Q201=-15 ;DEPTH ~	
	Q206=+250 ;FEED RATE FOR PLNGNG ~	
	Q202=+4 ;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	

13.8 OCM cycles for figure definition

13.8.1 Overview

OCM figures

Cycle	Call	Further information
1271 OCM RECTANGLE (#167 / #1-02-1) ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	**DEF**-active	Page 425
1272 OCM CIRCLE (#167 / #1-02-1) ■ Definition of a circle ■ Input of the circle diameter	**DEF**-active	Page 428
1273 OCM SLOT / RIDGE (#167 / #1-02-1) ■ Definition of a slot or ridge ■ Input of the width and the length	**DEF**-active	Page 430
1274 OCM CIRCULAR SLOT (#167 / #1-02-1) ■ Definition of a circular slot ■ Input of the width, the pitch circle, and the number of repeats	**DEF**-active	Page 434
1278 OCM POLYGON (#167 / #1-02-1) ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	**DEF**-active	Page 438
1281 OCM RECTANGLE BOUNDARY (#167 / #1-02-1) ■ Definition of a bounding rectangle	**DEF**-active	Page 441
1282 OCM CIRCLE BOUNDARY (#167 / #1-02-1) ■ Definition of a bounding circle	**DEF**-active	Page 443

13.8.2 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

Related topics

■ OCM cycles

Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626

Requirement

■ Software option Optimized Contour Machining (OCM (#167 / #1-02-1))

Description of function

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.

The control provides the following cycles for figure definition:

- **1271 OCM RECTANGLE**, see Page 425
- **1272 OCM CIRCLE**, see Page 428
- **1273 OCM SLOT / RIDGE**, see Page 430
- **1274 OCM CIRCULAR SLOT**, see Page 434
- **1278 OCM POLYGON**, see Page 438

The control provides the following cycles for figure boundary definition:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 441
- **1282 OCM CIRCLE BOUNDARY**, see Page 443

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**
1274 OCM CIRCULAR SLOT	**Q219 SLOT WIDTH**
1278 OCM POLYGON	**Q571 REF-CIRCLE DIAMETER**

You can define the following tolerances:

Tolerances	Example	Manufacturing dimension
DIN EN ISO 286-2	**10H7**	**10.0075**
DIN ISO 2768-1	**10m**	**10.0000**
Nominal dimension	**10+0.01-0.015**	**9.9975**

You can enter nominal dimensions with the following tolerances:

Combination	Example	Manufacturing dimension
a+-b	**10+-0.5**	**10.0**
a-+b	**10-+0.5**	**10.0**
a-b+c	**10-0.1+0.5**	**10.2**
a+b-c	**10+0.1-0.5**	**9.8**
a+b+c	**10+0.1+0.5**	**10.3**
a-b-c	**10-0.1-0.5**	**9.7**
a+b	**10+0.5**	**10.25**
a-b	**10-0.5**	**9.75**

Proceed as follows:

▶ Start the cycle definition

▶ Define the cycle parameters

▶ Select **NAME** in the action bar

▶ Enter a nominal dimension including tolerance

> ⓘ ▪ The control produces the workpiece to comply with the mean tolerance value.
> ▪ If you program a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
> ▪ Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

13.8.3 Cycle 1271 OCM RECTANGLE (#167 / #1-02-1)

ISO programming

G1271

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
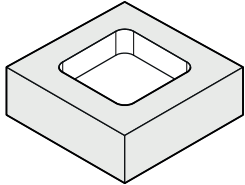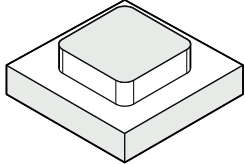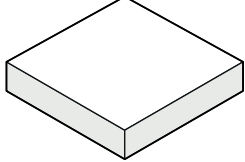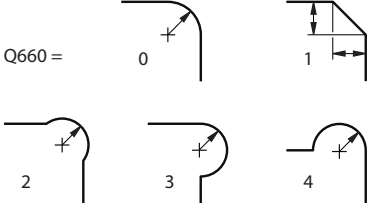
Notes

- 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**.

Notes on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

Cycle parameters

Help graphic	Parameter

Q650 = 0
  Q650 = 1
  Q650 = 2
  Q660 =
	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 424 Input: **0...99999.9999**
	Q219 Second side length? Length of the second 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 424 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. This value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	Q203 Workpiece surface coordinate?

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This 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 machining plane which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance in depth which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q260 Clearance height?

Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q578 Radius factor on inside corners?

The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path.

This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**.

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

13.8.4 Cycle 1272 OCM CIRCLE (#167 / #1-02-1)

ISO programming

G1272

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
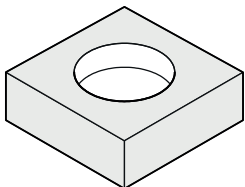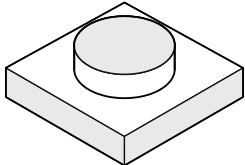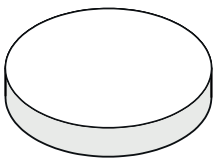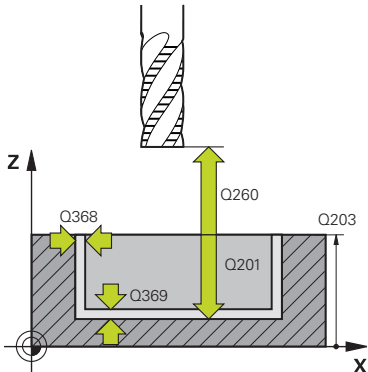
Notes

- 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**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

Cycle parameters

Help graphic	Parameter
Q650 = 0	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
Q650 = 1	**Q223 Circle diameter?** Diameter of the finished circle. You can program a tolerance if needed. **Further information:** "Tolerances", Page 424 Input: **0...99999.9999**
Q650 = 2	**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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners?

Help graphic	Parameter
	The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path.
	This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**.
	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

13.8.5 Cycle 1273 OCM SLOT / RIDGE (#167 / #1-02-1)

ISO programming

G1273

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
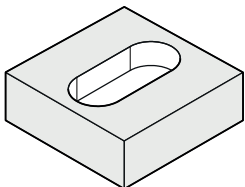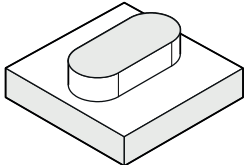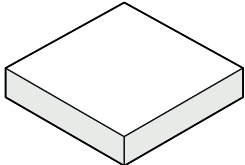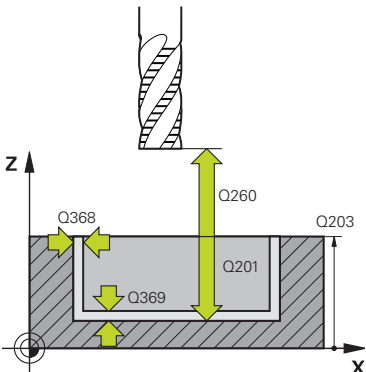
Notes

- 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**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

Cycle parameters

Help graphic	Parameter
Q650 = 0	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
Q650 = 1	**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 424 Input: **0...99999.9999**
Q650 = 2	**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 424 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. This value has an absolute effect. Input: **-360.000...+360.000**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor?

Help graphic	Parameter
	Finishing allowance in depth which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q260 Clearance height?
	Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners?
	The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path.
	This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**.
	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

13.8.6 Cycle 1274 OCM CIRCULAR SLOT (#167 / #1-02-1)

ISO programming

G1274

Application

Use figure cycle **1274 OCM CIRCULAR SLOT** to program a circular slot. Optionally, you can program a tolerance for the slot width.

When using Cycle **1274**, program the cycles in the following sequence:

- Cycle **1274 OCM CIRCULAR SLOT**
- Cycle **272 OCM ROUGHING**
- Cycle **273**, if required **OCM FINISHING FLOOR**
- Cycle **274**, if required **OCM FINISHING SIDE**
- Cycle **277**, if required **OCM CHAMFERING**

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1274** is DEF-active, which means that Cycle **1274** becomes active as soon as it has been defined in the NC program.
- The machining data defined in Cycle **1274** are valid for the OCM machining cycles **272** to **274** and **277**.

Notes on programming

- This cycle requires pre-positioning, which depends on the setting in parameter **Q367 REF. SLOT POSITION**.
- Make sure to define the angle between the starting point and the end point **Q248** in such a way that the contour does not intersect itself. Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	Q219 Width of slot? Slot width This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 424 Input: **0...99999.9999**
	Q375 Pitch circle diameter? The pitch circle diameter is the center line path of the slot. Input: **0...99999.9999**
	Q376 Starting angle? Polar angle of starting point This value has an absolute effect. Input: **-360.000...+360.000**
	Q248 Angular length? The opening angle is the angle between the starting point and the end point of the circular slot. This value has an incremental effect. Input: **0...360**
	Q378 Intermediate stepping angle? Angle between two machining positions The center of rotation is at the center of the slot. This parameter is effective when the number of machining operations is **Q377>=2**. 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**
	Q367 Ref. for slot pos. (0/1/2/3)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool position = center of the pitch circle **1**: Tool position = center of left figure arc **2**: Tool position = center of figure center axis **3**: Tool position = center of right figure arc Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999** **Q369 Finishing allowance for floor?** Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999** **Q260 Clearance height?** Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF** **Q578 Radius factor on inside corners?** The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. Input: **0.05...0.99**

Example

11 CYCL DEF 1274 OCM CIRCULAR SLOT ~	
Q219=+10	;SLOT WIDTH ~
Q375=+60	;PITCH CIRCLE DIAMETR ~
Q376=+0	;STARTING ANGLE ~
Q248=+60	;ANGULAR LENGTH ~
Q378=+90	;STEPPING ANGLE ~
Q377=+4	;NR OF REPETITIONS ~
Q367=+0	;REF. SLOT POSITION ~
Q203=+0	;SURFACE COORDINATE ~
Q201=-20	;DEPTH ~
Q368=+0.1	;ALLOWANCE FOR SIDE ~
Q369=+0.1	;ALLOWANCE FOR FLOOR ~
Q260=+100	;CLEARANCE HEIGHT ~
Q578=+0.2	;INSIDE CORNER FACTOR

13.8.7 Cycle 1278 OCM POLYGON (#167 / #1-02-1)

ISO programming

G1278

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
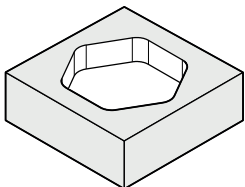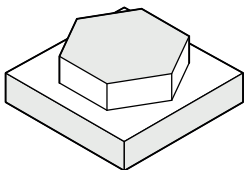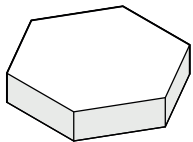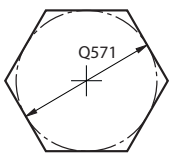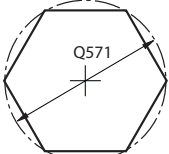
Notes

- 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**.
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

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 424

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. This 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. This 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 machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for inter-mediate positions and when retracting at the end of the cycle. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path. This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**. 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

13.8.8 Cycle 1281 OCM RECTANGLE BOUNDARY (#167 / #1-02-1)

ISO programming

G1281
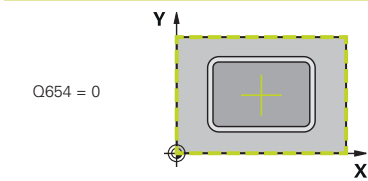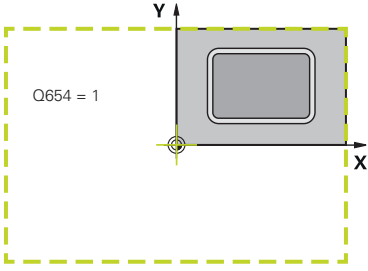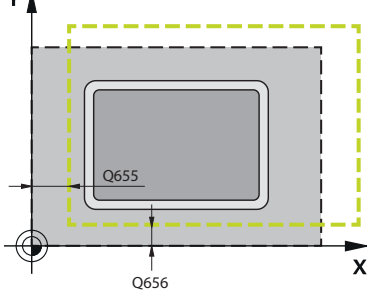
Application

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.

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 **1274** 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. This value has an incremental effect. Input: **0.001...9999.999**
	Q652 Length of minor axis? Length of the second side of the boundary, parallel to the secondary axis. This value has an incremental effect. 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

13.8.9 Cycle 1282 OCM CIRCLE BOUNDARY (#167 / #1-02-1)

ISO programming

G1282

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.

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 **1274** and **1278**.

Cycle parameters

Help graphic	Parameter

Q654 = 0

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**

Q654 = 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

14

Cycles for Drilling, Centering and Thread Machining

14.1 Overview

The control offers the following cycles for all types of drilling operations:

Drilling

Cycle		Call	Further information
200	**DRILLING**	**CALL**-active	Page 448
	■ Basic hole		
	■ Input of the dwell time at top and bottom		
	■ Depth reference selectable		
201	**REAMING**	**CALL**-active	Page 452
	■ Reaming a hole		
	■ Input of the dwell time at bottom		
202	**BORING**	**CALL**-active	Page 454
	■ Boring a hole		
	■ Input of the retraction feed rate		
	■ Input of the dwell time at bottom		
	■ Input of the retracting movement		
203	**UNIVERSAL DRILLING**	**CALL**-active	Page 458
	■ Degression – hole with decreasing infeed		
	■ Input of the dwell time at top and bottom		
	■ Input of chip breaking behavior		
	■ Depth reference selectable		
205	**UNIVERSAL PECKING**	**CALL**-active	Page 464
	■ Degression – hole with decreasing infeed		
	■ Input of chip breaking behavior		
	■ Input of a deepened starting point		
	■ Input of an advanced stop distance		
208	**BORE MILLING**	**CALL**-active	Page 472
	■ Milling of a hole		
	■ Input of a pre-drill diameter		
	■ Climb or up-cut milling selectable		
241	**SINGLE-LIP D.H.DRLNG**	**CALL**-active	Page 477
	■ 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		

Countersinking and centering

Cycle		Call	Further information
204	**BACK BORING**	**CALL**-active	Page 487
	■ Machining a counterbore on the underside of the workpiece		
	■ Input of the dwell time		
	■ Input of the retracting movement		

Cycle		Call	Further information
240	**CENTERING**	**CALL**-active	Page 491
	■ Drilling a center hole		
	■ Input of the centering diameter or depth		
	■ Input of the dwell time at bottom		

Tapping

Cycle		Call	Further information
18	**THREAD CUTTING**	**CALL**-active	Page 494
	■ With controlled spindle		
	■ Spindle stops at the bottom of the hole		
206	**TAPPING**	**CALL**-active	Page 497
	■ With a floating tap holder		
	■ Input of the dwell time at bottom		
207	**RIGID TAPPING**	**CALL**-active	Page 500
	■ Without a floating tap holder		
	■ Input of the dwell time at bottom		
209	**TAPPING W/ CHIP BRKG**	**CALL**-active	Page 505
	■ Without a floating tap holder		
	■ Input of chip breaking behavior		

Thread milling

Cycle		Call	Further information
262	**THREAD MILLING**	**CALL**-active	Page 511
	■ Milling a thread into pre-drilled material		
263	**THREAD MLLNG/CNTSNKG**	**CALL**-active	Page 516
	■ Milling a thread into pre-drilled material		
	■ Machining a countersunk chamfer		
264	**THREAD DRILLNG/MLLNG**	**CALL**-active	Page 521
	■ Drilling into solid material		
	■ Milling a thread		
265	**HEL. THREAD DRLG/MLG**	**CALL**-active	Page 526
	■ Milling a thread into solid material		
267	**OUTSIDE THREAD MLLNG**	**CALL**-active	Page 530
	■ Milling an external thread		
	■ Machining a countersunk chamfer		

14.2 Drilling

14.2.1 Cycle 200 DRILLING

ISO programming
G200

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

Related topics

■ Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 458

■ Cycle **205 UNIVERSAL PECKING** optionally with with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

Further information: "Cycle 205 UNIVERSAL PECKING ", Page 464

■ Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole

Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 477

Cycle run

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! There is a danger of collision!
>
> ▶ 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 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.

> ℹ 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. This 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**
	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

Help graphic	Parameter
	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	

14.2.2 Cycle 201 REAMING

ISO programming

G201

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! There is a danger of collision!
▶ 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 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

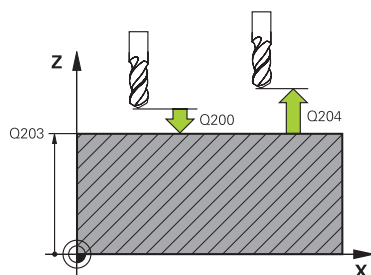| Help graphic	Parameter
	Q200 Set-up clearance?
	Distance between tool tip and workpiece surface. This value has an incremental effect.
	Input: **0...99999.9999** or **PREDEF**
	Q201 Depth?
	Distance between workpiece surface and bottom of hole. This value has an incremental effect.
	Input: **-99999.9999...+99999.9999**
	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. This 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	

14.2.3 Cycle 202 REAMING

ISO programming

G202

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

<div>

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! There is a danger of collision!

▶ 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

</div>

<div>

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.

</div>

<div>

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

</div>

- 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

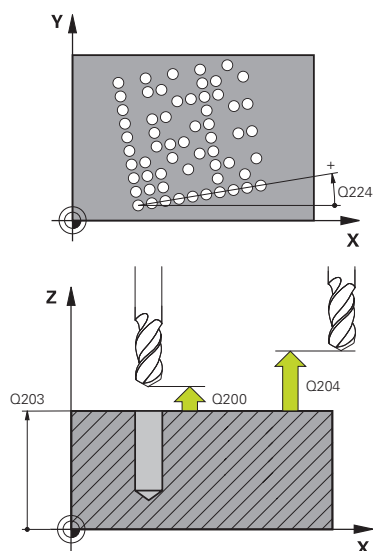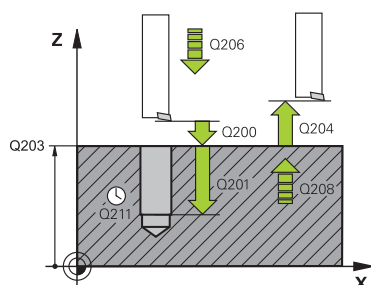| Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	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. This 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. This value has an absolute effect. Input: **0...360**
	Q357 Safety clearance to the side? Distance between tool tooth and the wall. This value has an incremental effect.

Help graphic	Parameter
	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	

14.2.4 Cycle 203 UNIVERSAL DRILLING

ISO programming

G203

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.

Related topics

■ Cycle **200 DRILLING** for simple holes

Further information: "Cycle 200 DRILLING", Page 448

■ Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

Further information: "Cycle 205 UNIVERSAL PECKING ", Page 464

■ Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole

Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 477

Cycle run

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 breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and, if necessary, remains there for the **DWELL TIME AT TOP Q210**

6   This sequence will be repeated until the **DEPTH Q201** is reached.

7   When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **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 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 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 the **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 the **SET-UP CLEARANCE Q200** or to the **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 **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** minus **DECREMENT Q212** at **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 the **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 the **SET-UP CLEARANCE Q200** or to the **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! There is a danger of collision!
>
> ▶ 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 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. This 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 PLUNGING 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**

Q205 Minimum plunging depth?

Help graphic	Parameter
	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

ISO programming

G205

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 470

Related topics

■ Cycle **200 DRILLING** for simple holes
  Further information: "Cycle 200 DRILLING", Page 448

■ Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
  Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 458

■ Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
  Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 477

Cycle run

1   The control positions the tool in the tool axis at **FMAX** to the entered **SET-UP 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! There is a danger of collision!

▶ 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 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
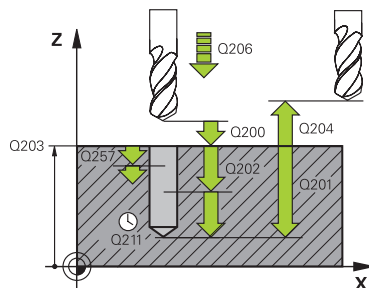
Help graphic	Parameter

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. This 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**

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?

Help graphic	Parameter
	Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after the 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 break-ing. 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 incre-mental 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**
	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**

Help graphic	Parameter
	>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
9 M30	; End of program
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 **Q201 DEPTH** 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	
9 M30	; End of program	
10 END PGM 205 MM		

14.2.6   Cycle 208 BORE MILLING

ISO programming

G208

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 run

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**

> **ℹ** 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:

Tolerances	Example	Manufacturing dimension
DIN EN ISO 286-2	**10H7**	**10.0075**
DIN ISO 2768-1	**10m**	**10.0000**
Nominal dimension	**10+0.01-0.015**	**9.9975**

You can enter nominal dimensions with the following tolerances:

Combination	Example	Manufacturing dimension
a+-b	**10+-0.5**	**10.0**
a-+b	**10-+0.5**	**10.0**
a-b+c	**10-0.1+0.5**	**10.2**
a+b-c	**10+0.1-0.5**	**9.8**
a+b+c	**10+0.1+0.5**	**10.3**
a-b-c	**10-0.1-0.5**	**9.7**
a+b	**10+0.5**	**10.25**
a-b	**10-0.5**	**9.75**

Proceed as follows:

▶ Start the cycle definition

▶ Define the cycle parameters

▶ Select **NAME** in the action bar

▶ Enter a nominal dimension including tolerance

> ⓘ
> ■ The control produces the workpiece to comply with the mean tolerance value.
> ■ If you program a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
> ■ Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

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! There is a danger of collision!

▶ 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. This 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. This value has an absolute effect. You can program a tolerance if needed.

Further information: "Tolerances", Page 473

Input: **0...99999.9999**

Q342 Roughing diameter?

Enter the dimension of the pre-drilled diameter. This value has an absolute effect.

Input: **0...99999.9999**

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?

Help graphic	Parameter
	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	

14.2.7 Cycle 241 SINGLE-LIP D.H.DRLNG

ISO programming

G241

Application

Cycle **241 SINGLE-LIP D.H.DRLNG** machines holes with a single-lip deep hole drill. It is possible to enter a recessed starting point. The control performs moving to drilling depth with **M3**. You can change the direction of rotation and the rotational speed for moving into and retracting from the hole.

Related topics

■ Cycle **200 DRILLING** for simple holes

Further information: "Cycle 200 DRILLING", Page 448

■ Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking

Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 458

■ Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance

Further information: "Cycle 205 UNIVERSAL PECKING ", Page 464

Cycle run

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

2 Depending on the positioning behavior, the control will either switch on the spindle with the programmed speed at the **SET-UP CLEARANCE Q200** or at a certain distance above the coordinate surface.

Further information: "Position behavior when working with Q379", Page 483

3 The control executes the approach motion depending on the definition of **Q426 DIR. OF SPINDLE ROT.** with a spindle that rotates clockwise, counterclockwise, or is stationary

4 The tool drills with **M3** and **Q206 FEED RATE FOR PLNGNG** to the drilling depth **Q201** or dwell depth **Q435** or the plunging depth **Q202**:

■ After defining **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after reaching the dwell depth and remains there for **Q211 DWELL TIME AT DEPTH**

■ If a smaller infeed value has been entered, the control drills to the plunging depth. The plunging depth is decreased after each infeed by **Q212 DECREMENT**

5 If programmed, the tool remains at the hole bottom for chip breaking.

6 After the control has reached the hole depth, it will automatically switch off the coolant, set the speed to the value defined in **Q427 ROT.SPEED INFEED/OUT** and, if required, change again the direction of rotation from **Q426**.

7 The control positions the tool to the retract position at **Q208 RETRACTION FEED RATE**.

Further information: "Position behavior when working with Q379", Page 483

8 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! There is a danger of collision!
>
> ▶ 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. This 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**

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.

Help graphic	Parameter
	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: **1...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 482
	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 **Q201 DEPTH**.
	"...": 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 482
	Input: **0...999**
	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?

Help graphic	Parameter
	Value by which the control decreases **Q202 PLUNGING 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

A 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 NC function **CALL PGM**, for example. You define a macro as an NC program with the file type *.h or *.i.

- 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 1006

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 (for example, 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 of drilling is 0.4 mm or inch above the recessed starting point. So if the recessed starting point is at −2, the control starts the drilling process at −1.6 mm.

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 recessed start point. So if the recessed 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 Countersinking and centering

14.3.1 Cycle 204 BACK BORING

ISO programming
G204

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.
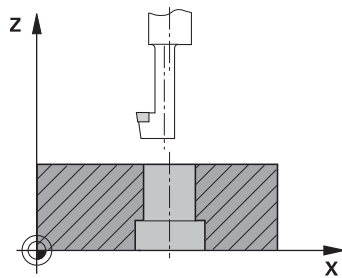
Cycle sequence

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

<table>
<tr><td align="center">*NOTICE*</td></tr>
<tr><td>

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.

</td></tr>
</table>

■ 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.

> **ℹ** 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
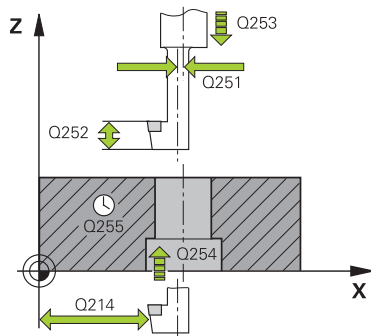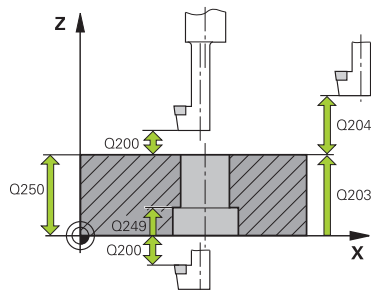
Help graphic	Parameter

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. This 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 offsets the tool by the off-center distance (after orienting the spindle). Inputting 0 is not permitted

Help graphic	Parameter
	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 This 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	

14.3.2 Cycle 240 CENTERING

ISO programming

G240

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

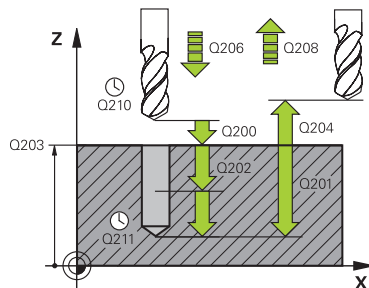| *NOTICE* |
|---|
| **Danger of collision!** |
| If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision! |
| ▶ 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 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.
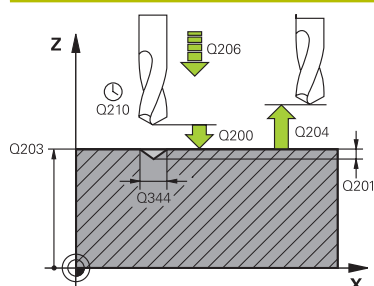
Cycle parameters

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. This 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**
	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.

Help graphic	Parameter
	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		

14.4 Tapping

14.4.1 Cycle 18 THREAD CUTTING

ISO programming
G86

Application
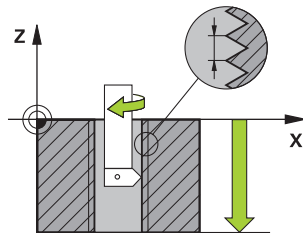
Cycle **18 THREAD CUTTING** moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.

Related topics

■ Cycles for Thread Machining

Notes

⚙️ Cycle **18 THREAD CUTTING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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.
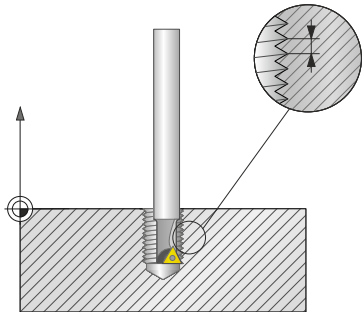
Notes on programming

- ■ 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

14.4.2 Cycle 206 TAPPING

ISO programming

G206

Application

The thread is cut in one or more passes. A floating tap holder is used.

Related topics

- Cycle **207 RIGID TAPPING** without floating tap holder

  Further information: "Cycle 207 RIGID TAPPING ", Page 500

- Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking

  Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 505

Cycle run

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! There is a danger of collision!
>
> ▶ 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. This 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)

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

↑ Tool Retract	▶ Select **Tool Retract**
[NC Start icon]	▶ Press the **NC Start** key
	> The tool retracts from the hole and moves to the starting point of machining.
	> The spindle is stopped automatically. The control issues an error message.
	▶ Cancel the NC program with the **INTERNAL STOP** button
	or
	▶ Acknowledge the error message and continue with **NC Start**

ⓘ ■ **Program Run** operating mode:

When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.

■ **MDI** application:

When you call a thread cycle, the **Tool Retract** button appears. The button is grayed out until you press **NC stop**.

14.4.3 Cycle 207 RIGID TAPPING

ISO programming

G207

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.

Related topics

■ Cycle **206 TAPPING** with floating tap holder

Further information: "Cycle 206 TAPPING ", Page 497

■ Cycle **209 TAPPING W/ CHIP BRKG** without floating tap holder, but optionally with chip breaking

Further information: "Cycle 209 TAPPING W/ CHIP BRKG ", Page 505

Cycle run

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

> **i** 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

⚙️   Cycle **207 RIGID TAPPING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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! There is a danger of collision!

▶ 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.

ℹ️   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. This 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	

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

▶ Select **Tool Retract**

▶ Press the **NC Start** key
> The tool retracts from the hole and moves to the starting point of machining.
> The spindle is stopped automatically. The control issues an error message.
▶ Cancel the NC program with the **INTERNAL STOP** button

or

▶ Acknowledge the error message and continue with **NC Start**

ℹ
- **Program Run** operating mode:
  When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:
  When you call a thread cycle, the **Tool Retract** button appears. The button is grayed out until you press **NC stop**.

14.4.4 Cycle 209 TAPPING W/ CHIP BRKG

ISO programming

G209

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.

Related topics
- Cycle **206 TAPPING** with floating tap holder
  Further information: "Cycle 206 TAPPING ", Page 497
- Cycle **207 RIGID TAPPING** without floating tap holder
  Further information: "Cycle 207 RIGID TAPPING ", Page 500

Cycle run

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

⚙   Cycle **209 TAPPING W/ CHIP BRKG** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

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! There is a danger of collision!

▶   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.

ℹ   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. This 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. This value has an absolute effect.

Input: **0...360**

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.

Help graphic	Parameter
	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	

Retraction with stopped NC program

You can retract a thread-turning tool as follows in stopped state:

 ▶ Select **Tool Retract**

 ▶ Press the **NC Start** key
> The tool retracts from the hole and moves to the starting point of machining.
> The spindle is stopped automatically. The control issues an error message.
▶ Cancel the NC program with the **INTERNAL STOP** button
  or
▶ Acknowledge the error message and continue with **NC Start**

ⓘ ■ **Program Run** operating mode:

When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.

■ **MDI** application:

When you call a thread cycle, the **Tool Retract** button appears. The button is grayed out until you press **NC stop**.

14.5 Thread milling

14.5.1 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

> **ℹ** 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.

14.5.2 Cycle 262 THREAD MILLING

ISO programming

G262

Application

With this cycle, you can mill a thread into pre-drilled material.

Related topics

- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer
  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 516

- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 521

- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer
  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 526

- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer
  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 530

Cycle run

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

> ℹ 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! There is a danger of collision!
>
> ▶ 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance?

Help graphic	Parameter
	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	

14.5.3 Cycle 263 THREAD MLLNG/CNTSNKG

ISO programming

G263

Application

With this cycle, you can mill a thread into pre-drilled material. In addition, you can use it to machine a countersunk chamfer.

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material

  Further information: "Cycle 262 THREAD MILLING ", Page 511

- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 521

- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer

  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 526

- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 530

Cycle run

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! There is a danger of collision!

▶ 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.
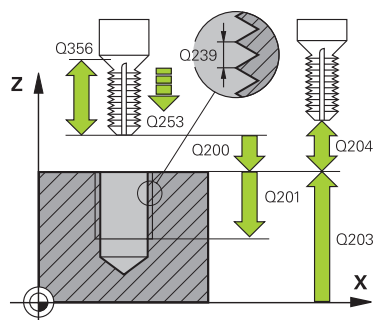
Cycle parameters

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**
	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.

Help graphic	Parameter
	Input: **0...99999.9999**
	Q203 Workpiece surface coordinate?
	Coordinate on the workpiece surface referenced to the active datum. This 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	

14.5.4 Cycle 264 THREAD DRILLNG/MLLNG

ISO programming

G264

Application

With this cycle, you can drill into solid material, machine a counterbore, and finally mill a thread.

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material

  Further information: "Cycle 262 THREAD MILLING ", Page 511

- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer

  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 516

- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer

  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 526

- Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 530

Cycle run

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

<div style="border:1px solid #000;">

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! There is a danger of collision!

▶ 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

</div>

- 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.

> ℹ Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.
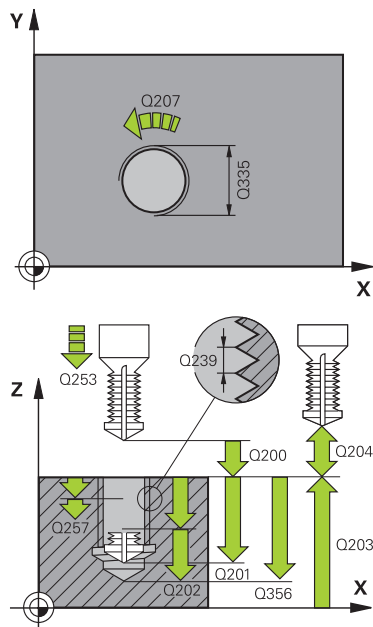
Cycle parameters

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**
	Q257 Infeed depth for chip breaking?

Help graphic	Parameter
	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. This 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	

14.5.5   Cycle 265 HEL. THREAD DRLG/MLG

ISO programming

G265

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.

Related topics

■ Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material

Further information: "Cycle 262 THREAD MILLING ", Page 511

■ Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer

Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 516

■ Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer

Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 521

■ Cycle **267 OUTSIDE THREAD MLLNG** for milling an external thread, optionally machining of a countersunk chamfer

Further information: "Cycle 267 OUTSIDE THREAD MLLNG ", Page 530

Cycle run

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! There is a danger of collision!
▶ 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. This 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	

14.5.6 Cycle 267 OUTSIDE THREAD MLLNG

ISO programming

G267

Application

With this cycle, you can mill an external thread. In addition, you can use it to machine a countersunk chamfer.

Related topics

- Cycle **262 THREAD MILLING** for milling a thread into pre-drilled material

  Further information: "Cycle 262 THREAD MILLING ", Page 511

- Cycle **263 THREAD MLLNG/CNTSNKG** for milling a thread into pre-drilled material, optionally machining of a countersunk chamfer

  Further information: "Cycle 263 THREAD MLLNG/CNTSNKG ", Page 516

- Cycle **264 THREAD DRILLNG/MLLNG** for drilling into solid material and milling a thread, optionally machining of a countersunk chamfer

  Further information: "Cycle 264 THREAD DRILLNG/MLLNG ", Page 521

- Cycle **265 HEL. THREAD DRLG/MLG** for milling a thread into solid material, optionally machining of a countersunk chamfer

  Further information: "Cycle 265 HEL. THREAD DRLG/MLG ", Page 526

Cycle run

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! There is a danger of collision!

▶ 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
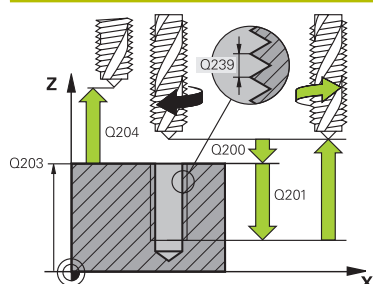
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

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**
	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?

Help graphic	Parameter
	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. This 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

15

Milling Cycles

15.1   Overview

Pocket milling

Cycle		Call	Further information
251	**RECTANGULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	**CALL**-active	Page 539
252	**CIRCULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical	**CALL**-active	Page 545
253	**SLOT MILLING** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL**-active	Page 552
254	**CIRCULAR SLOT** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL**-active	Page 558

Stud milling

Cycle		Call	Further information
256	**RECTANGULAR STUD** ■ Roughing and finishing cycle ■ Approach position: selectable	**CALL**-active	Page 565
257	**CIRCULAR STUD** ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter	**CALL**-active	Page 572
258	**POLYGON STUD** ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter	**CALL**-active	Page 577

Milling contours with SL cycles

Cycle		Call	Further information
20	**CONTOUR DATA** ■ Input of machining information	**DEF**-active	Page 588
21	**PILOT DRILLING** ■ Machining a hole for non-center cutting tools	**CALL**-active	Page 590
22	**ROUGH-OUT** ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	**CALL**-active	Page 592
23	**FLOOR FINISHING** ■ Finishing with finishing allowance for the floor from Cycle **20**	**CALL**-active	Page 597

Cycle		Call	Further information
24	**SIDE FINISHING** ■ Finishing with side finishing allowance from Cycle **20**	**CALL**-active	Page 600
270	**CONTOUR TRAIN DATA** ■ Input of contour data for Cycle **25** or **276**	**DEF**-active	Page 603
25	**CONTOUR TRAIN** ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	**CALL**-active	Page 605
275	**TROCHOIDAL SLOT** ■ Machining of open and closed slots using trochoidal milling	**CALL**-active	Page 610
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 616

Milling contours with OCM Cycles

Cycle		Call	Further information
271	**OCM CONTOUR DATA** (#167 / #1-02-1) ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	**DEF**-active	Page 631
272	**OCM ROUGHING** (#167 / #1-02-1) ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	**CALL**-active	Page 633
273	**OCM FINISHING FLOOR** (#167 / #1-02-1) ■ 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 639
274	**OCM FINISHING SIDE** (#167 / #1-02-1) ■ Finishing with side finishing allowance from Cycle **271**	**CALL**-active	Page 642
277	**OCM CHAMFERING** (#167 / #1-02-1)	**CALL**-active	Page 644

Cycle	Call	Further information
■ Deburr the edges		
■ Consideration of adjacent contours and walls		

Milling planes

Cycle		Call	Further information
232	**FACE MILLING**	**CALL**-active	Page 661
	■ Face mill a level surface in multiple infeeds		
	■ Selection of the milling plan		
233	**FACE MILLING**	**CALL**-active	Page 669
	■ Roughing and finishing cycle		
	■ Roughing strategy and direction: selectable		
	■ Input of side walls		

Engraving

Cycle		Call	Further information
225	**ENGRAVING**	**CALL**-active	Page 681
	■ Engrave texts on a plane surface		
	■ Arranged in a straight line or along a circular arc		

15.2 Milling pockets

15.2.1 Cycle 251 RECTANGULAR POCKET

ISO programming
G251

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! There is a danger of collision!

▶ 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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 545

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
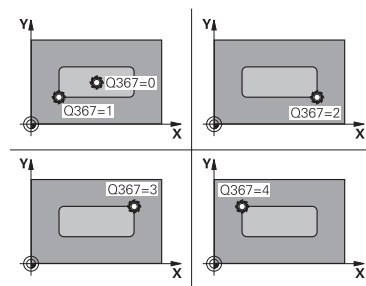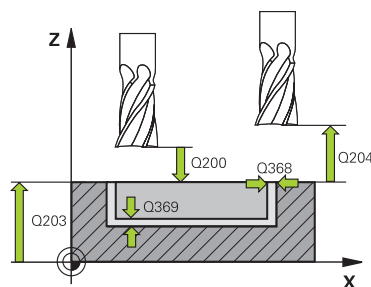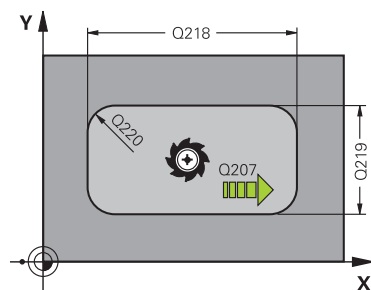
Help graphic	Parameter
	Q215 Machining operation (0/1/2)?
	Define the machining operation:
	0: Roughing and finishing
	1: Only roughing
	2: Only finishing
	Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined
	Input: **0**, **1**, **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 machining plane which remains after roughing. 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. This 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 in depth which remains after roughing. 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?

Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.

0: Finishing in one infeed

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. This 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...1.41** or **PREDEF**

Q366 Plunging strategy (0/1/2)?

Help graphic	Parameter
	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**
	Further information: "Plunging strategy Q366 with RCUTS", Page 545
	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:

$$Helical\,radius = R_{corr} - RCUTS$$

  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.

15.2.2 Cycle 252 CIRCULAR POCKET

ISO programming
G252

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! There is a danger of collision!

▶ 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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 552
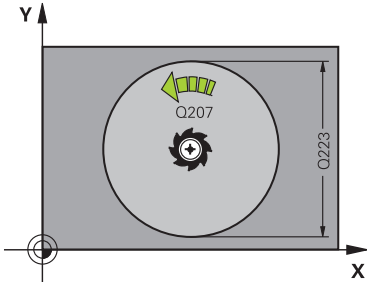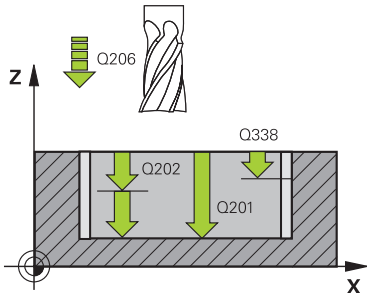
Notes on programming

- 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 machining plane which remains after roughing. 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 in depth which remains after roughing. 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?

Help graphic	Parameter
	Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.
	0: Finishing in one infeed
	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. This 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?
	Q370x 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**
	Further information: "Plunging strategy Q366 with RCUTS", Page 552
	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

Help graphic	Parameter
	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:

$$Helical radius = R_{corr} - RCUTS$$

  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.

15.2.3 Cycle 253 SLOT MILLING

ISO programming

G253

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! There is a danger of collision!

▶ 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! There is a danger of collision!

▶ 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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
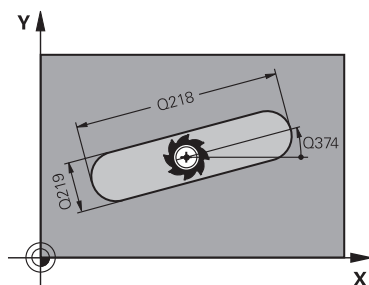
Help graphic	Parameter

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. This value has an incremental effect.

Input: **0...99999.9999**

Q219 Width of slot?

Enter the width of the slot, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. This value has an incremental effect.

Maximum slot width for roughing: Twice the tool diameter

Input: **0...99999.9999**

Q368 Finishing allowance for side?

Finishing allowance in the machining plane which remains after roughing. 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. This value has an absolute effect.

Input: **-360.000...+360.000**
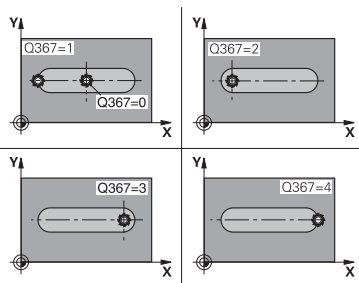
Q367 Position of slot (0/1/2/3/4)?

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**

Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

+1 = climb milling
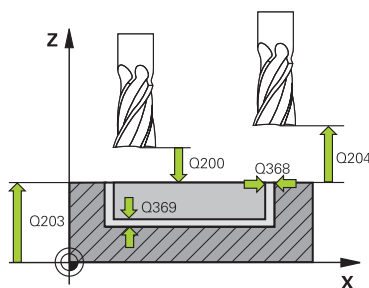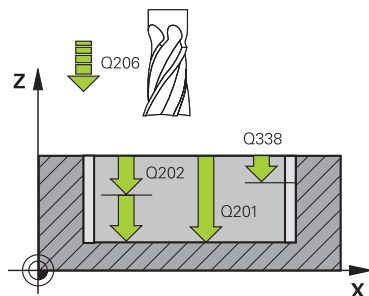
Help graphic	Parameter
	-1 = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF** **Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. 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 in depth which remains after roughing. 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? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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. This 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** **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. Alternative: **PREDEF** Input: **0**, **1**, **2**

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 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	

15.2.4 Cycle 254 CIRCULAR SLOT

ISO programming

G254

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! There is a danger of collision!

▶ 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! There is a danger of collision!

▶ 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. This value has an incremental effect.
	Maximum slot width for roughing: Twice the tool diameter
	Input: **0...99999.9999**
	Q368 Finishing allowance for side?
	Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q375 Pitch circle diameter?
	The pitch circle diameter is the center line path of the slot.
	Input: **0...99999.9999**
	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**. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis?

Help graphic	Parameter
	Center of the pitch circle in the secondary axis of the working plane. **Only effective if Q367 = 0**. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q376 Starting angle?
	Polar angle of starting point
	Input: **-360.000...+360.000**
	Q248 Angular length?
	The opening angle is the angle between the starting point and the end point of the circular slot. This value has an incremental effect.
	Input: **0...360**
	Q378 Intermediate stepping angle?
	Angle between two machining positions
	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 in depth which remains after roughing. 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?

Help graphic	Parameter
	Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.
	0: Finishing in one infeed
	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. This 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**
	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

The help graphic shows a cross-section diagram with axes z and x, illustrating Q200, Q368, Q204, Q369, and Q203 dimensions relative to tools and workpiece.

Help graphic	Parameter
	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	

15.3 Milling studs

15.3.1 Cycle 256 RECTANGULAR STUD

ISO programming
G256

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! There is a danger of collision!

▶ 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.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- 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

Help graphic

Q218 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**

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. This value has an incremental effect.

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. This value has an incremental effect.

Input: **0...99999.9999**

Q425 Workpiece blank side length 2?

Length of stud blank parallel to the secondary axis of the working plane. This value has an incremental effect.

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 machining plane which remains after roughing. 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. This 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. This 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?

Help graphic	Parameter
	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 in depth which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q338 Infeed for finishing?

Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.

0: Finishing in one infeed

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	

15.3.2 Cycle 257 CIRCULAR STUD

ISO programming

G257

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! There is a danger of collision!
>
> ▶ 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.

■ This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

■ 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
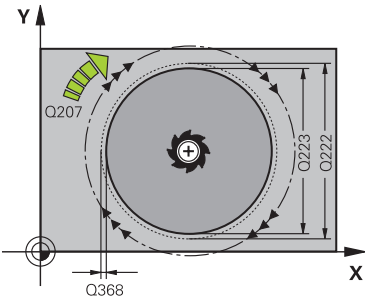
Help graphic	Parameter
	Q223 Finished part diameter? Diameter of the finished stud Input: **0...99999.9999**
	Q222 Workpiece blank diameter? Diameter of workpiece blank. The workpiece blank diameter must be greater than the diameter of the finished part. The control performs multiple stepovers if the 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 machining plane which remains 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**
	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?

Help graphic	Parameter
	Coordinate on the workpiece surface referenced to the active datum. This 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 in depth which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q338 Infeed for finishing?
	Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.
	0: Finishing in one infeed
	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 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	

15.3.3 Cycle 258 POLYGON STUD

ISO programming

G258

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! There is a danger of collision!

▶ 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 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. There is a danger of collision!

▶ 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 absolute coordinates (no 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.

■ This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

■ 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. This 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**
	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?

Help graphic	Parameter
	Finishing allowance in depth which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q338 Infeed for finishing?
	Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.
	0: Finishing in one infeed
	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	

15.3.4 Programming examples

Example: Milling pockets, studs and slots
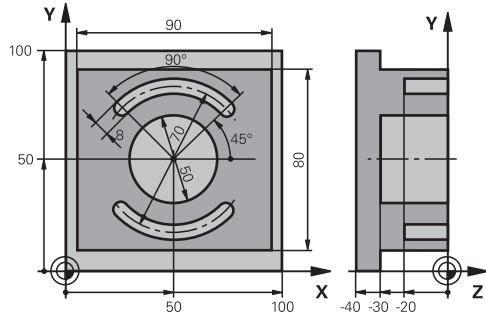
0	BEGIN PGM C210 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL CALL 6 Z S3500	; Tool call: roughing/finishing
4	L Z+100 R0 FMAX M3	; Retract the tool
5	CYCL DEF 256 RECTANGULAR STUD ~	
	Q218=+90 ;FIRST SIDE LENGTH ~	
	Q424=+100 ;WORKPC. BLANK SIDE 1 ~	
	Q219=+80 ;2ND SIDE 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
14 M30	; End of program
15 END PGM C210 MM	

15.4 Milling contours with SL cycles

15.4.1 Fundamentals

Application

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**.

> **i** Instead of SL cycles, HEIDENHAIN recommends the powerful Optimized Contour Milling function software option (#167 / #1-02-1).

Related topics

- Optimized contour milling (#167 / #1-02-1)

  Further information: "Milling contours with OCM cycles (#167 / #1-02-1)", Page 626
- Contour call with a simple contour formula **CONTOUR DEF**

  Further information: "Simple contour formula", Page 380
- Contour call with a complex contour formula **SEL CONTOUR**

  Further information: "Complex contour formula", Page 383
- Contour call with cycle **14 CONTOUR**

  Further information: "Cycle 14 CONTOUR ", Page 379

Description of function

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

0 BEGIN SL 2 MM
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM

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.

15.4.2   Cycle 20 CONTOUR DATA

ISO programming

G120

Application

Use Cycle **20** to specify machining data for the subprograms describing the subcontours.

Related topics

- Cycle **271 OCM CONTOUR DATA** (#167 / #1-02-1)

  Further information: "Cycle 271 OCM CONTOUR DATA (#167 / #1-02-1) ", Page 631

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
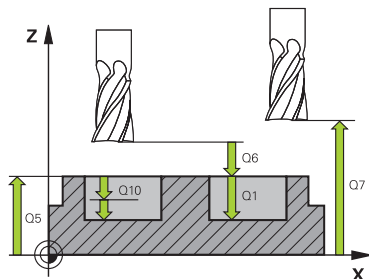
Help graphic	Parameter
	Q1 Milling depth? Distance between workpiece surface and pocket floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q2 Path overlap factor? **Q2** x tool radius = stepover factor k Input: **0.0001...1.9999**
	Q3 Finishing allowance for side? Finishing 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). This 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**
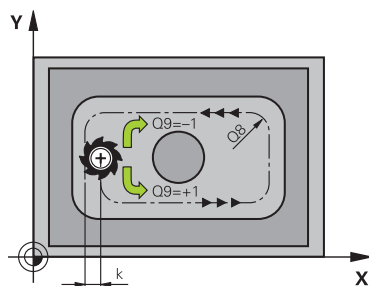
Example

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

15.4.3 Cycle 21 PILOT DRILLING

ISO programming

G121

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).
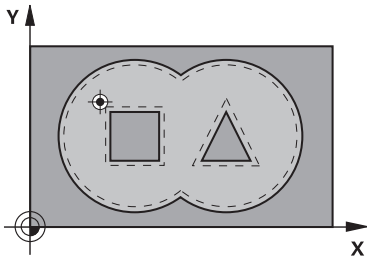
Notes

- 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

15.4.4 Cycle 22 ROUGH-OUT

ISO programming

G122

Application
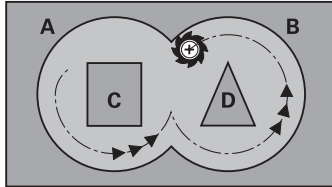
Use Cycle **22 ROUGH-OUT** to define the technology data for roughing.

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

Related topics

- Cycle **272 OCM ROUGHING** (#167 / #1-02-1)
  Further information: "Cycle 272 OCM ROUGHING (#167 / #1-02-1)", Page 633

Cycle run

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. There is a danger of collision!
>
> ▶ 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.

■ The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

Further information: "Adapting the feed rate for circular paths with M109", Page 976

> **i** 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 can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting 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**
	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 moves the tool during fine roughing:

Help graphic	Parameter
	0: Between areas that need to be fine-roughed, the control moves the tool along the contour at the current depth. The entry is effective only when the diameter of the fine-roughing tool is larger than or equal to the coarse roughing tool radius.
	1: Between the areas that need to be fine-roughed, the control retracts the tool to the 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

15.4.5 Cycle 23 FLOOR FINISHING

ISO programming

G123

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

Related topics

- Cycle **273 OCM FINISHING FLOOR** (#167 / #1-02-1)

  Further information: "Cycle 273 OCM FINISHING FLOOR (#167 / #1-02-1)", Page 639

Cycle run

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. There is a danger of collision!

▶ 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 976

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
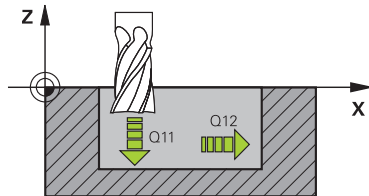
Help graphic	Parameter
	Q11 Feed rate for plunging?
	Tool traversing speed in mm/min during plunging
	Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing?
	Traversing feed rate in the working plane
	Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate 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

15.4.6 Cycle 24 SIDE FINISHING

ISO programming

G124

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** if required **ROUGH-OUT**

Related topics

■ Cycle **274 OCM FINISHING SIDE** (#167 / #1-02-1)

Further information: "Cycle 274 OCM FINISHING SIDE (#167 / #1-02-1)", Page 642

Cycle run

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).

> 🛈 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. There is a danger of collision!
>
> ▶ 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 976

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
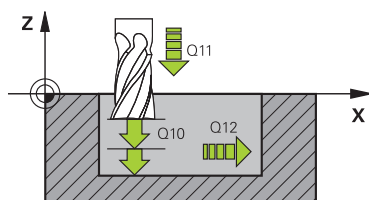
Help graphic	Parameter
	Q9 Direction of rotation? cw = -1 Machining direction: **+1**: Counterclockwise **−1**: Clockwise Input: **-1**, **+1**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing 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

15.4.7 Cycle 270 CONTOUR TRAIN DATA

ISO programming
G270

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 takes effect 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	Parameter
	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

15.4.8 Cycle 25 CONTOUR TRAIN

ISO programming
G125

Application
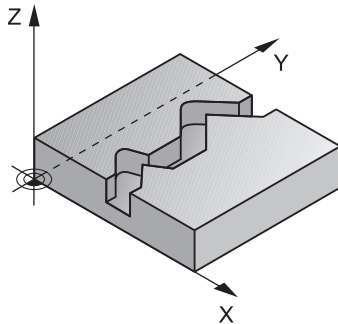
In conjunction with Cycle **14 CONTOUR**, this cycle enables you to machine open and closed contours.

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. There is a danger of collision!

▶ 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.
  Further information: "Adapting the feed rate for circular paths with M109", Page 976

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). This 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 can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting 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 recip-rocating 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**.

Help graphic	Parameter
	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

15.4.9 Cycle 275 TROCHOIDAL SLOT

ISO programming

G275

Application

In conjunction with Cycle **14 CONTOUR**, 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** (#45 / #2-31-1)).

Further information: "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

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. There is a danger of collision!

▶ 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**.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 976

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, which must be parallel to the secondary axis of the working plane. If the slot width equals the tool diameter, the control will mill an oblong hole. This value has an incremental effect.
	Maximum slot width for roughing: Twice the tool diameter
	Input: **0...99999.9999**
	Q368 Finishing allowance for side?
	Finishing allowance in the machining plane which remains after roughing. 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. This 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**
	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?

Help graphic	Parameter
	Traversing speed of the tool in mm/min for moving to depth
	Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing?
	Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.
	0: Finishing in one infeed
	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. This 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**
	Q369 Finishing allowance for floor?
	Finishing allowance in depth which remains after roughing. 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

Help graphic	Parameter
	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	

15.4.10 Cycle 276 THREE-D CONT. TRAIN

ISO programming
G276

Application
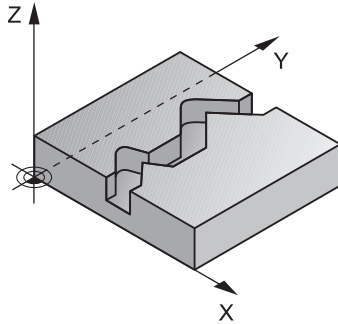
In conjunction with Cycle **14 CONTOUR** and Cycle **270 CONTOUR TRAIN DATA**, this cycle enables you to machine open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete for example inside corners with a smaller tool.

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 run

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. There is a danger of collision!

- ▶ 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.
- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 976

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). This 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 can use the action bar selection to apply the coarse roughing tool directly from the tool table. In addition, you can enter the tool name yourself by selecting 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?

Help graphic	Parameter
	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

15.4.11 Programming examples

Example: Roughing-out and fine-roughing a pocket with SL Cycles
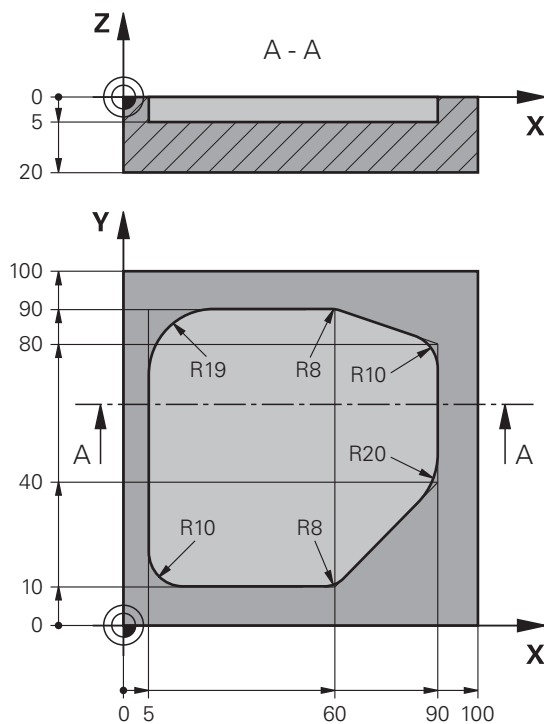
0 BEGIN PGM 1078634 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 15 Z S4500	; Tool call: coarse roughing tool (diameter: 30)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-5 ;MILLING 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
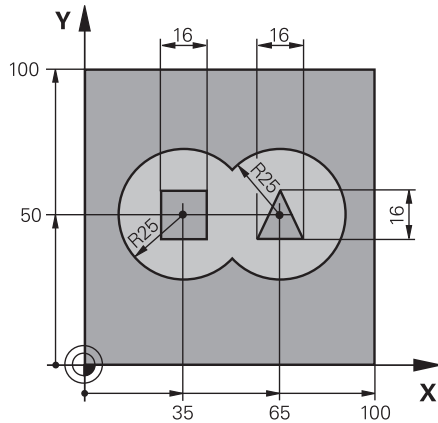
0 BEGIN PGM 2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 204 Z S2500	; Tool call: drill (diameter: 12)
4 L Z+250 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1 /2 /3 /4	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20 ;MILLING DEPTH ~	
Q2=+1 ;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
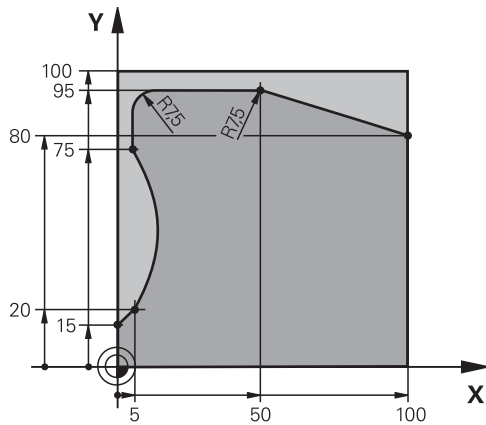
0 BEGIN PGM 3 MM		
1 BLK FORM 0.1 Z X+0 Y+0 Z-40		
2 BLK FORM 0.2 X+100 Y+100 Z+0		
3 TOOL CALL 10 Z S2000	; Tool call (diameter: 20)	
4 L Z+100 R0 FMAX M3	; Retract the tool	
5 CYCL DEF 14.0 CONTOUR		
6 CYCL DEF 14.1 CONTOUR LABEL1		
7 CYCL DEF 25 CONTOUR TRAIN ~		
Q1=-20 **;MILLING DEPTH ~**		
Q3=+0 **;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	
10 M30	; End of program	
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	

15.5 Milling contours with OCM cycles (#167 / #1-02-1)

15.5.1 Fundamentals

Application

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 are more powerful than Cycles **22** to **24**. OCM cycles provide 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.

Related topics

- Contour call with a simple contour formula **CONTOUR DEF**
  Further information: "Simple contour formula", Page 380
- Contour call with a complex contour formula **SEL CONTOUR**
  Further information: "Complex contour formula", Page 383
- OCM cycles for figure definition
  Further information: "OCM cycles for figure definition", Page 422

Description of function

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 figure 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**

▶ Program cycle parameter **Q569=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 cycle 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 by 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 figure definition", Page 422

Removing residual material

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to remove the residual material. During finishing the control will take into account the material roughed out, thus preventing the finishing tool from being overloaded.

Further information: "Example: Open pocket and fine roughing with OCM cycles", Page 648

> **ⓘ** ■ If residual material remains in the inside corners after roughing, then use a smaller rough-out tool or define an additional roughing operation with a smaller tool.
> ■ If the inside corners cannot be roughed out completely, the control may damage the contour during chamfering. In order to prevent damage to the contour, follow the procedure described below.

Procedure regarding residual material in inside corners

The example describes the inside machining of a contour by using several tools with radii greater than the programmed contour. Although the radius of the tools used becomes smaller, residual material remains in the inside corners after roughing. The control takes this residual material into account during the subsequent finishing and chamfering operations.

In the example, you use the following tools:

■ **MILL_D20_ROUGH**, Ø 20 mm
■ **MILL_D10_ROUGH**, Ø 10 mm
■ **MILL_D6_FINISH**, Ø 6 mm
■ **NC_DEBURRING_D6**, Ø 6 mm
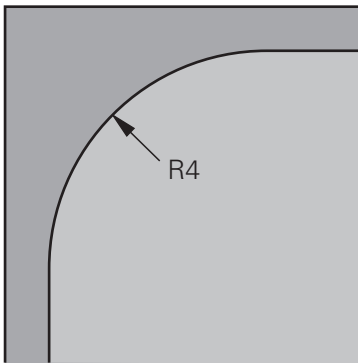
Inside corner with a radius of 4 mm in this example

Roughing

▶ Rough the contour with the tool **MILL_D20_ROUGH**

> The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 12 mm during initial roughing.

...	
12 TOOL CALL Z "MILL_D20_ROUGH"	
...	
15 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR**	Resulting inside radius = R_T+ (**Q578** * R_T) **10 + (0.2 *10) = 12**
...	
16 CYCL DEF 272 OCM ROUGHING	
...	

▶ Then rough the contour with the smaller tool **MILL_D10_ROUGH**

> The control takes into account the Q parameter **Q578 INSIDE CORNER FACTOR**, resulting in inside radii of 6 mm during initial roughing.

...	
20 TOOL CALL Z "MILL_D10_ROUGH"	
...	
22 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR**	Resulting inside radius = R_T+ (**Q578** * R_T) **5 + (0.2 *5) = 6**
...	
23 CYCL DEF 272 OCM ROUGHING	
... **Q438 = -1 ;ROUGH-OUT TOOL**	**-1**: The control assumes that the tool last used is the rough-out tool
...	

Finishing

▶ Finish the contour with the tool **MILL_D6_FINISH**

> This finishing tool would allow inside radii of 3.6 mm. This means that the finishing tool would be capable of machining the defined inside radii of 4 mm. However, the control takes into account the residual material of the rough-out tool **MILL_D10_ROUGH**. The control machines the contour with the previous roughing tool's inside radii of 6 mm. Thus, the finishing cutter will be protected from overload.

...	
27 TOOL CALL Z "MILL_D6_FINISH"	
...	
29 CYCL DEF 271 OCM CONTOUR DATA	
... **Q578 = 0.2 ;INSIDE CORNER FACTOR**	Resulting inside radius = R_T+ (**Q578** * R_T) **3 + (0.2 *3) = 3.6**
...	
30 CYCL DEF 274 OCM FINISHING SIDE	
... **Q438 = -1 ;ROUGH-OUT TOOL**	**-1**: The control assumes that the tool last used is the rough-out tool
...	

Chamfering

▶ Chamfering the contour: When defining the cycle, you must define the last rough-out tool of the roughing operation.

> 🛈 If you use the finishing tool as a roughing tool, the control will damage the contour. In this case, the control assumes that the finishing cutter machined the contour with inside radii of 3.6 mm. However, the finishing cutter has limited the inside radii to 6 mm based on the previous roughing operation.

...	
33 TOOL CALL Z "NC_DEBURRING_D6"	
...	
35 CYCL DEF 277 OCM CHAMFERING	
...	Rough-out tool of the last roughing operation
QS438 = "MILL_D10_ROUGH" ;ROUGH-OUT TOOL	
...	

Positioning logic in OCM cycles

The current tool position is above the clearance height:

1 The control moves the tool to the starting point in the working plane at rapid traverse.

2 The tool moves at **FMAX** to **Q260 CLEARANCE HEIGHT** and then to **Q200 SET-UP CLEARANCE**

3 The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**.

The current tool position is below the clearance height:

1 The control moves the tool to **Q260 CLEARANCE HEIGHT** at rapid traverse.

2 At **FMAX**, the tool moves to the starting point in the working plane and then to **Q200 SET-UP CLEARANCE**

3 The control then positions the tool to the starting point in the tool axis at **Q253 F PRE-POSITIONING**

> 🛈 Programming and operating notes:
> - **Q260** The control uses the **CLEARANCE HEIGHT** from Cycle **271 OCM CONTOUR DATA** or from the figure cycles.
> - **Q260 CLEARANCE HEIGHT** is effective only when the position of the safe height is above the safety distance.

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.

Example

Program structure: Machining with OCM cycles

The table below shows an example of how a program run with the OCM cycles might look like.

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
...
35 CYCL DEF 277OCM CHAMFERING
36 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

15.5.2 Cycle 271 OCM CONTOUR DATA (#167 / #1-02-1)

ISO programming

G271

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. This 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 machining plane which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q369 Finishing allowance for floor?
	Finishing allowance in depth which remains after roughing. This value has an incremental effect.
	Input: **0...99999.9999**
	Q260 Clearance height?
	Position in the tool axis at which no collision can occur with the workpiece. The control approaches this position for intermediate positions and when retracting at the end of the cycle. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners?
	The tool radius multiplied with **Q578 INSIDE CORNER FACTOR** results in the smallest tool center point path.
	This prevents smaller inside radii at the contour, as resulting from the tool radius plus the product of tool radius and **Q578 INSIDE CORNER FACTOR**.
	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

15.5.3 Cycle 272 OCM ROUGHING (#167 / #1-02-1)

ISO programming

G272

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 (#167 / #1-02-1)", Page 1183

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 run

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

   Further information: "Positioning logic in OCM cycles", Page 630

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 634

   - **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**)

8 Finally, the tool moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 CLEARANCE HEIGHT**

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**
- R_{corr} (tool radius **R** + tool oversize **DR**)

Helical:

The helical path is calculated as follows:

$$Helical radius = R_{corr} - RCUTS$$

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** .
- If you have roughed a figure or a contour before, program the number or the name of the rough-out tool in the cycle. If there was no initial roughing, you need to define **Q438=0 ROUGH-OUT TOOL** in the cycle parameter during the first roughing operation.

Cycle parameters

Help graphic	Parameter
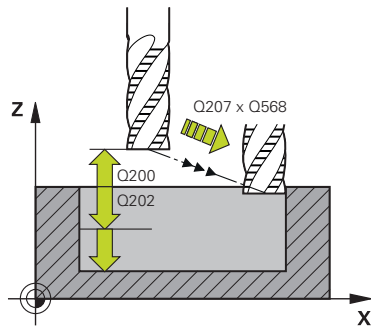
Q202 Plunging depth?

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

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

Help graphic	Parameter
	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**
	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**
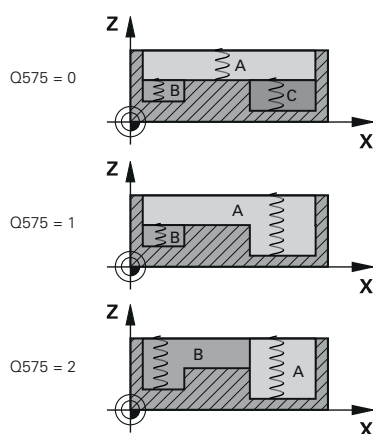
Help graphic	Parameter
	🛈 The total plunging path is the sum of all plunging movements.

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

15.5.4   Cycle 273 OCM FINISHING FLOOR (#167 / #1-02-1)

ISO programming

G273

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 run

1 The tool uses positioning logic to move to the starting point

   Further information: "Positioning logic in OCM cycles", Page 630

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 moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 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
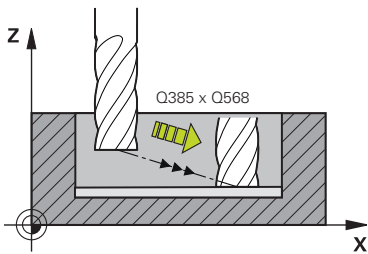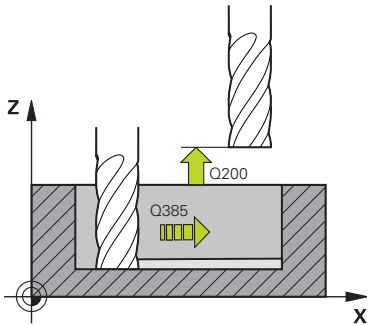
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.
- For finishing with Cycle **273**, the tool always works in climb milling mode.
- 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

Help graphic

Q595 = 0

Q595 = 1

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

15.5.5 Cycle 274 OCM FINISHING SIDE (#167 / #1-02-1)

ISO programming

G274

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 run

1 The tool uses positioning logic to move to the starting point

2 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

   Further information: "Positioning logic in OCM cycles", Page 630

3 The control then moves the tool to the first plunging depth using the feed rate for plunging

4 The tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately

5 Finally, the tool moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** 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 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.

■ The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

   Further information: "Adapting the feed rate for circular paths with M109", Page 976

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
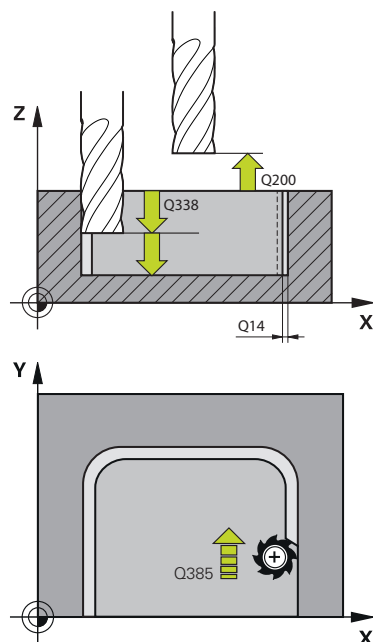
Help graphic	Parameter
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side 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

15.5.6 Cycle 277 OCM CHAMFERING (#167 / #1-02-1)

ISO programming

G277

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
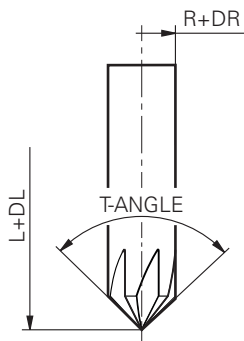
Before the control can execute Cycle **277**, you need to create the tool in the tool table using appropriate parameters:

- **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 run

1   The tool uses positioning logic to move to the starting point. This point is determined automatically based on the programmed contour

Further information: "Positioning logic in OCM cycles", Page 630

2   In the next step, the tool moves at **FMAX** to set-up clearance **Q200**

3   Then, the tool plunges vertically to **Q353 DEPTH OF TOOL TIP**

4   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**

5   Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).

6   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

7   Finally, the tool moves with **Q253 F PRE-POSITIONING** to **Q200 SET-UP CLEARANCE** and then at **FMAX** to **Q260 CLEARANCE HEIGHT**
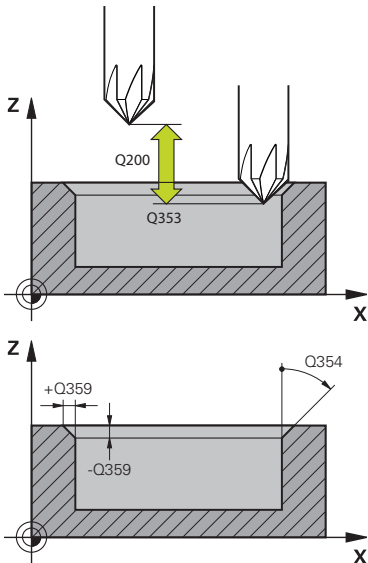
Notes

- 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.

- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** figure cycles will remain intact.

- The cycle monitors for damage to the contour floor from the tool tip. This tool tip results from the radius **R**, the radius of the tool tip **R_TIP**, and the point angle **T-ANGLE**.

- Keep in mind that the active tool radius of the chamfering tool must be smaller than 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. This tool radius results from **T-ANGLE** and **R_TIP** from the tool table.

- The cycle considers the miscellaneous functions **M109** and **M110**. During the inside and outside machining of circular arcs the control keeps the feed rate constant at the cutting edge for inside and outside radii.

  Further information: "Adapting the feed rate for circular paths with M109", Page 976

- If the roughing operations have not completely removed the material before chamfering, you need to define the last roughing tool in **QS438 ROUGH-OUT TOOL**, in order to prevent damage to the contour.

  "Procedure regarding residual material in inside corners"

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

15.5.7 Programming examples

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**
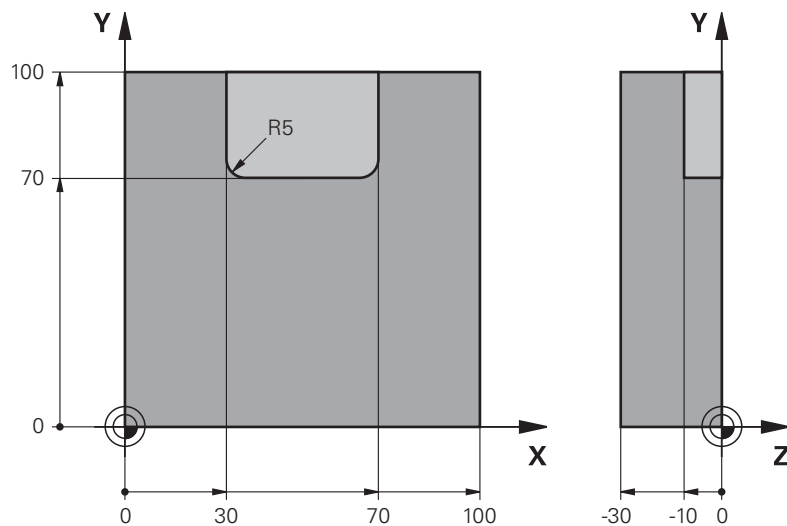
0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S8000 F1500	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;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**
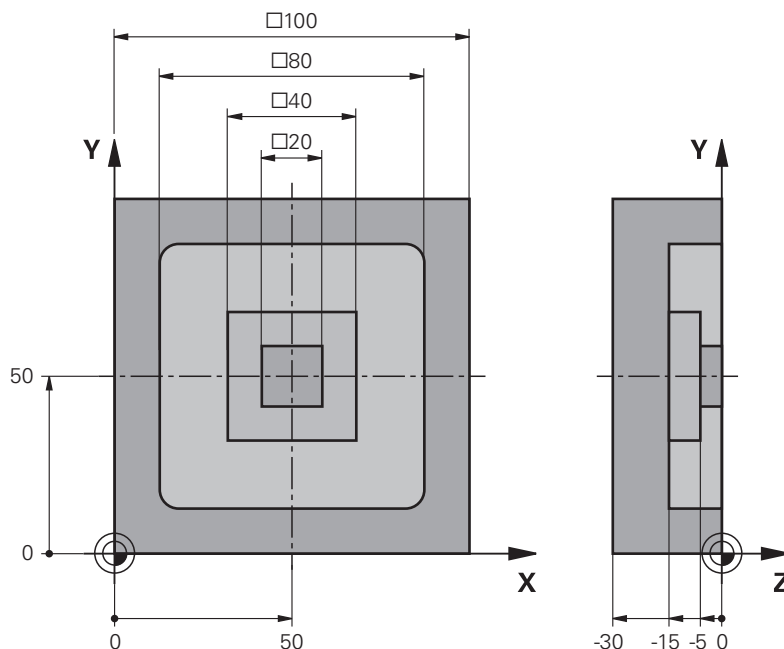
0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S8000 F1500	; Tool call (diameter: 10 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-15 ;DEPTH ~	
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 X+20	
33 L Y+20	
34 L X-20	
35 L Y-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
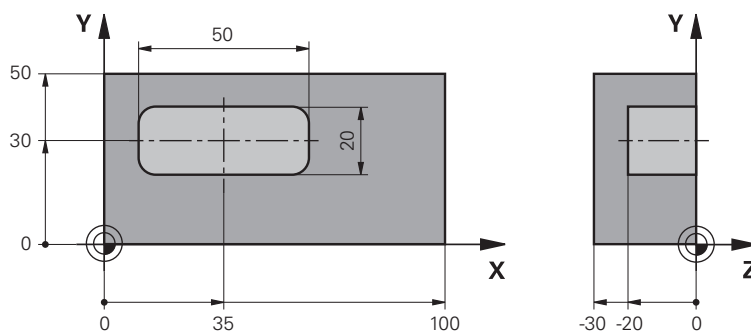
0 BEGIN PGM FACE_MILL MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+50 Z+2	
3 TOOL CALL 6 Z S5000 F3000	; Tool call (diameter: 12 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF P1 = LBL 1 I2 = LBL 1 DEPTH2 P3 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+2 ;SURFACE COORDINATE ~	
Q201=-22 ;DEPTH ~	
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=-0 ;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**
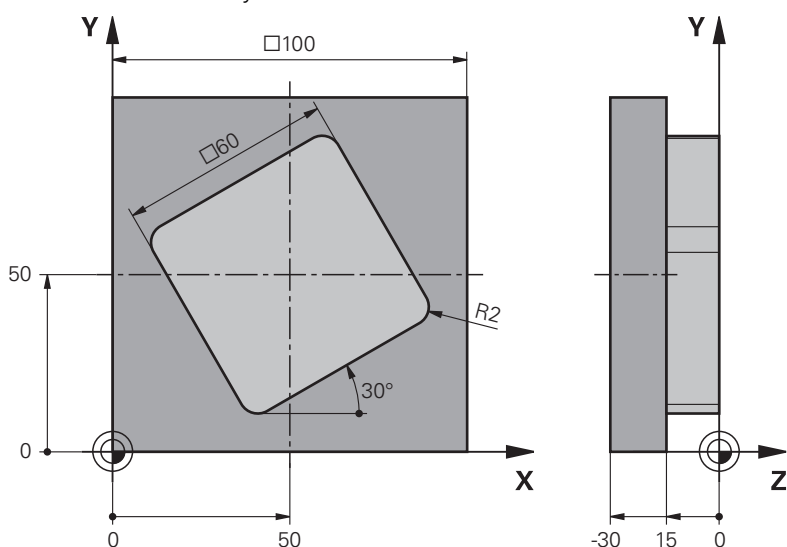
0 BEGIN PGM OCM_FIGURE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE ~	
Q650=+1 ;FIGURE TYPE ~	
Q218=+60 ;FIRST SIDE LENGTH ~	
Q219=+60 ;2ND SIDE LENGTH ~	
Q660=+0 ;CORNER 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=+0.4	;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: void areas with OCM cycles

The following NC program shows how to define void areas by using OCM cycles.
Two circles from the previous machining operation are used to define void areas in
CONTOUR DEF. The tool plunges perpendicularly within the void area.

Program sequence

- Tool call: drill (diameter: 20 mm)
- Define Cycle **200**
- Tool call: roughing cutter (diameter: 14 mm)
- Define **CONTOUR DEF** with void areas
- Define Cycle **271**
- Define and call Cycle **272**
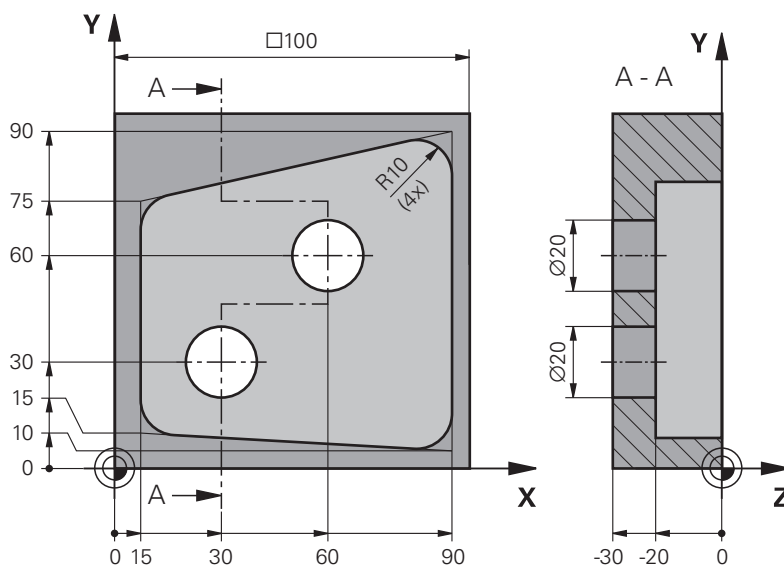
0 BEGIN PGM VOID_1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 206 Z S8000 F900	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-30 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q395=+1 ;DEPTH REFERENCE	
6 L X+30 Y+30 R0 FMAX M99	
7 L X+60 Y+60 R0 FMAX M99	
8 TOOL CALL 7 Z S7000 F2000	; Tool call (diameter: 14 mm)

9 L Z+100 R0 FMAX M3	
10 CONTOUR DEF P1 = LBL 1 V1 = LBL 2 V2 = LBL 3	; Definition of contour and void area
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	
12 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.441 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253=+750 ;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=+13626 ;SPINDLE SPEED ~	
Q579=+1 ;PLUNGING FACTOR S ~	
Q575=+2 ;INFEED STRATEGY	
13 CYCL CALL	
14 M30	; End of program
15 LBL 1	; Contour subprogram 1
16 L X+90 Y+50	
17 L Y+10	
18 RND R10	
19 L X+10 Y+15	
20 RND R10	
21 L Y+75	
22 RND R10	
23 L X+90 Y+90	
24 RND R10	
25 L Y+50	
26 LBL 0	
27 LBL 2	; Void area 1
28 CC X+30 Y+30	
29 L X+40 Y+30	
30 C X+40 Y+30 DR-	
31 LBL 0	
32 LBL 3	; Void area 2

33 CC X+60 Y+60	
34 L X+70 Y+60	
35 C X+70 Y+60 DR-	
36 LBL 0	
37 END PGM VOID_1 MM	

15.6 Milling planes

15.6.1 Cycle 232 FACE MILLING

ISO programming

G232

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

Related topics

- Cycle **233 FACE MILLING**

  Further information: "Cycle 233 FACE MILLING ", Page 669

Cycle run

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
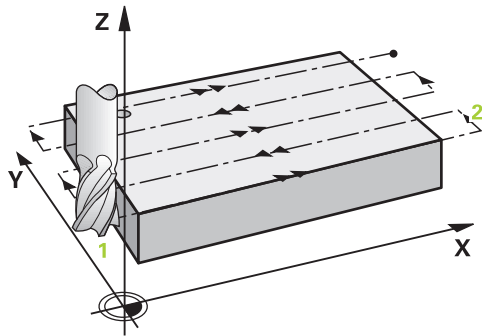
3   The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.

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
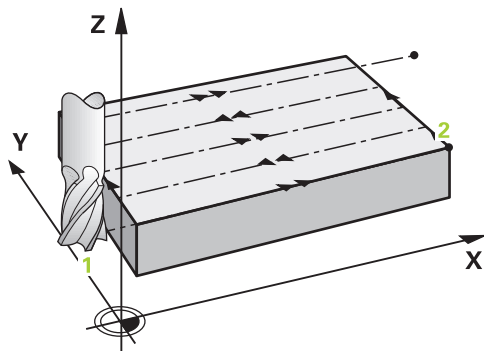
3   The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.

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. This 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. This 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. This 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. This 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?

Finishing allowance in depth which remains after roughing.

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**

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**.

Help graphic	Parameter
	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

15.6.2 Cycle 233 FACE MILLING

ISO programming

G233

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

Related topics

- Cycle **232 FACE MILLING**

  Further information: "Cycle 232 FACE MILLING ", Page 661

Strategies Q389=0 and Q389 =1
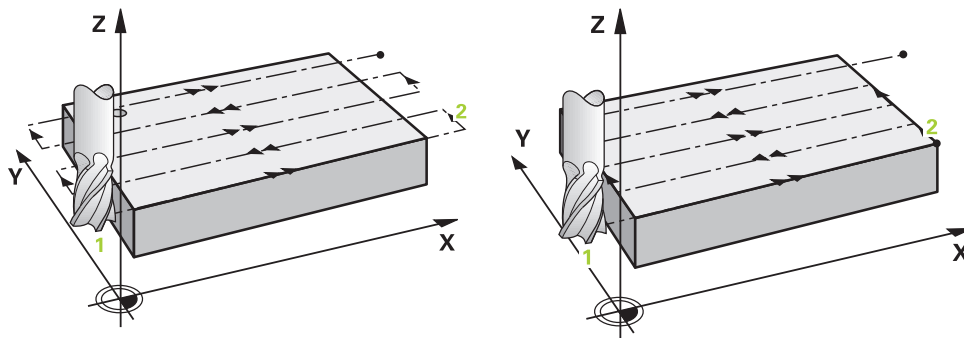
The strategies **Q389**=0 and **Q389**=1 differ in the overtravel during face milling. If **Q389**=0, the end point lies outside of the surface, with **Q389**=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the 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
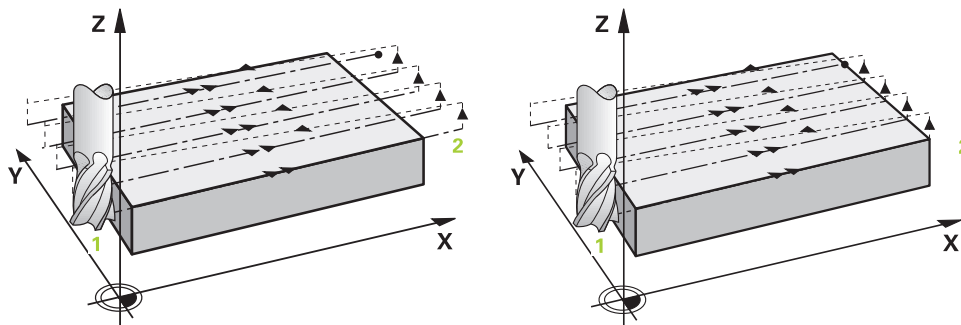
The strategies **Q389**=2 and **Q389**=3 differ in the overtravel during face milling. If **Q389**=2, the end point lies outside of the surface, with **Q389**=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the 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 **Q207** 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 **Q370** and the set-up clearance to the side **Q357**.

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—with lateral limitation

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:
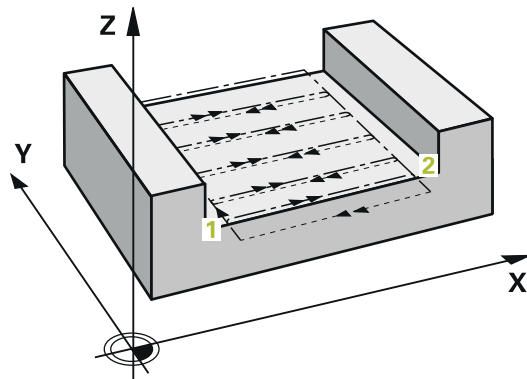
1   The control positions the tool at **FMAX** to the starting point in the working plane. This position is offset from the edge of the workpiece by the tool radius and the set-up clearance **Q357** to the side.

2   The tool moves at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** and from there at **Q207 FEED RATE MILLING** to the first plunging depth **Q202**.

3   The control moves the tool on a circular path to the starting 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 the **Q369 ALLOWANCE FOR FLOOR** at **Q385 FINISHING FEED RATE**.

9   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
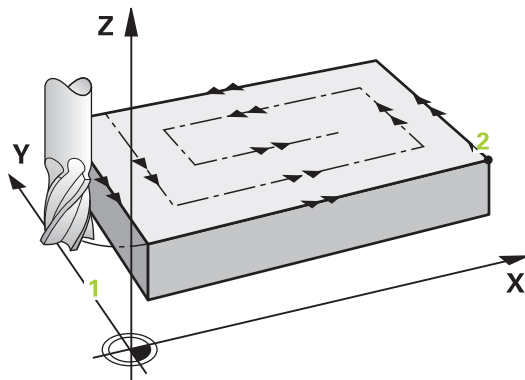
Cycle sequence

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.
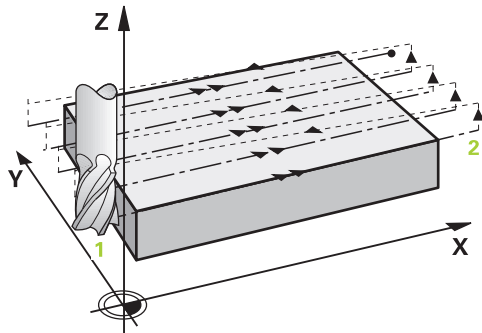
4   The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a tangential approach path.

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 completed. 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**.
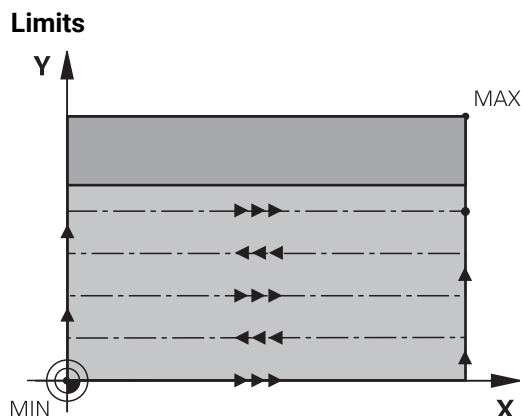
Limits

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! There is a danger of collision!

▶ 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.
■ This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

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.

> ℹ️ Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

Cycle parameters

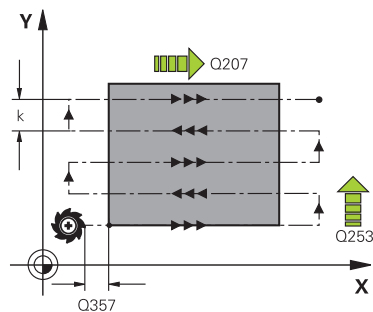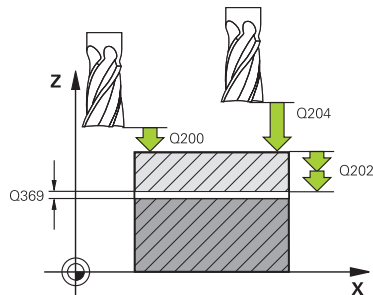| Help graphic	Parameter
	Q215 Machining operation (0/1/2)?
	Define the machining operation:
	0: Roughing and finishing
	1: Only roughing
	2: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined
	Input: **0**, **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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance in depth which remains after roughing. 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**
	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**
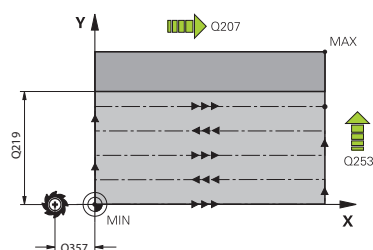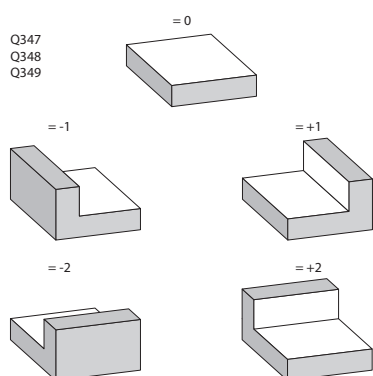
Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF** **Q204 2nd set-up clearance?** Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. 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** **Q368 Finishing allowance for side?** Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect. Input: **0...99999.9999** **Q338 Infeed for finishing?** Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect. **0:** Finishing in one infeed 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

Help graphic	Parameter
	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	

15.7 Engraving

15.7.1 Cycle 225 ENGRAVING

ISO programming
G225

Application
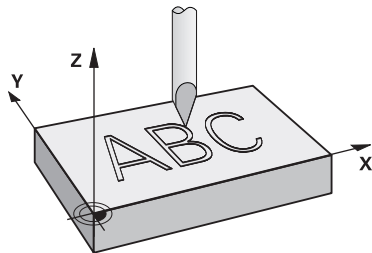
This cycle is used to engrave texts on a flat surface of the workpiece. You can arrange the texts in a straight line or along an arc.

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**.
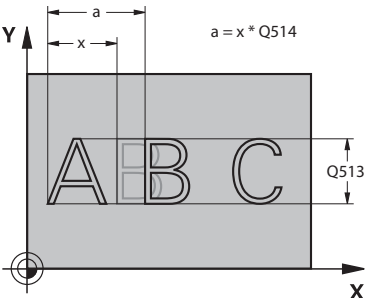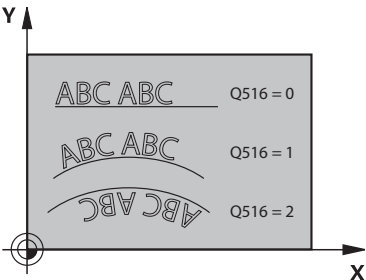
Notes

- 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 width of the characters varies. **X** = width of the character + default spacing. This factor allows you to influence the spacing. **Q514=0/1**: Default spacing between the characters **Q514>1**: The spacing between the characters is expanded. **Q514<1**: The spacing between the characters is reduced. This can lead to overlapping characters. Input: **0...10**
	Q515 Font? **0**: Font **DeJaVuSans** **1**: Font **LiberationSans-Regular** Input: **0**, **1**
	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**
	Q206 Feed rate for plunging?

Help graphic	Parameter
	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. This 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**
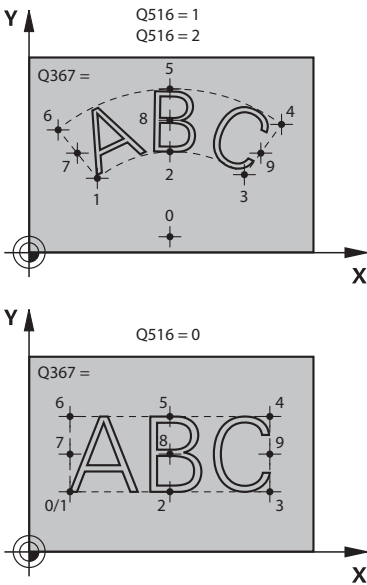
Help graphic	Parameter
	Q574 Maximum text length?
	Enter the maximum text length. The control also takes into account parameter **Q513** Character height.
	If **Q513 = 0**, the control engraves the text over exactly the length indicated in parameter **Q574**. The character height will be scaled accordingly.
	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**

ℹ️ 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	Character
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%at	@
%CE	CE

Non-printable characters

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).

> **i** Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9 (e.g., **%time08**).

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	ISO 8601 calendar week

> **i** 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 1058

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.

16

Coordinate Transformation

16.1 Reference systems

16.1.1 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 692
B-CS	Basic coordinate system basic coordinate system	Page 695
W-CS	Workpiece coordinate system workpiece coordinate system	Page 697
WPL-CS	Working plane coordinate system working plane coordinate system	Page 699
I-CS	Input coordinate system input coordinate system	Page 702
T-CS	Tool coordinate system tool coordinate system	Page 703

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.

16.1.2 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.
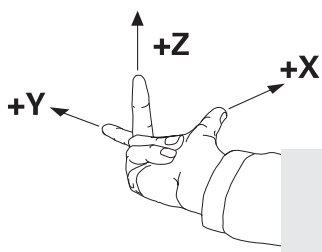
Origin of the coordinate system

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**.
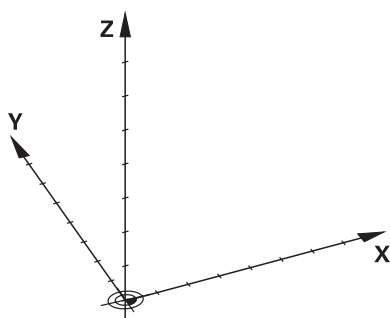
16.1.3 Machine coordinate system M-CS

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.
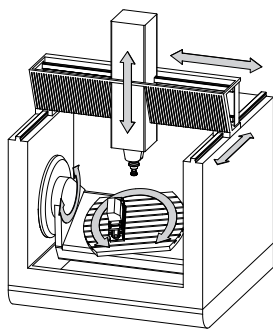
The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

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.
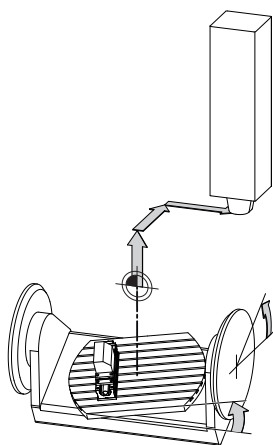
Position of the machine datum in the machine

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 *.pr", Page 1710

  > ⚙ The machine manufacturer configures the **OFFS** columns of the preset table in accordance with the machine.

- Axis-specific shifts in the rotary and parallel axes using the datum table

  Further information: "Datum table", Page 715
- Axis-specific shifts in the rotary and parallel axes using the **TRANS DATUM** function

  Further information: "Datum shift with TRANS DATUM", Page 727

> ⚙ The machine manufacturer can also define further transformations.
> **Further information:** "Note", Page 694

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 967

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 1635

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. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

---

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**.

16.1.4 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 705
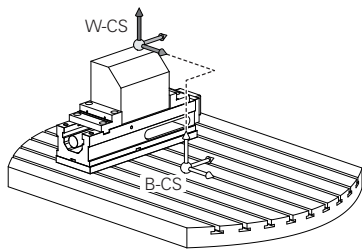
> ⚙ The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset table in accordance with the machine.
>
> **Further information:** "Note", Page 696

Note

The machine manufacturer can define additional basic transformations 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. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

---

16.1.5 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 705
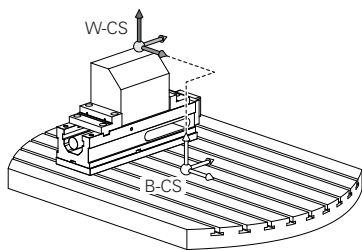
Transformations in the workpiece coordinate system (W-CS)

HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

- Axes **X**, **Y**, **Z** of the **TRANS DATUM** function before tilting the working plane

  Further information: "Datum shift with TRANS DATUM", Page 727
- Columns **X**, **Y**, **Z** of the datum table before tilting the working plane

  Further information: "Datum table", Page 715
- The **TRANS MIRROR** function or Cycle **8 MIRRORING** before tilting the working plane with spatial angles

  Further information: "Mirroring with TRANS MIRROR", Page 729

  Further information: "Cycle 8 MIRRORING", Page 717
- **PLANE** functions for tilting the working plane (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737

> **ℹ** You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

With these transformations, the position and orientation of the working plane coordinate system **WPL-CS** are changed.
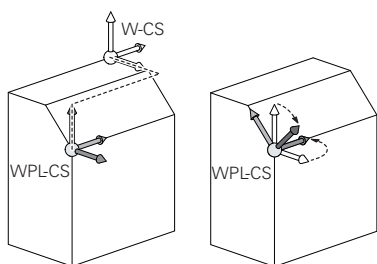
> ## *NOTICE*
>
> **Danger of collision!**
>
> The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.
>
> - ▶ Program only the recommended transformations in the respective 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.

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 702
- 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 699
- The result of transformations built upon each other depends on the programming sequence.

16.1.6 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 702

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 697

If no transformations are defined in the **W-CS**, then the position and orientation of the **W-CS** and **WPL-CS** are identical.
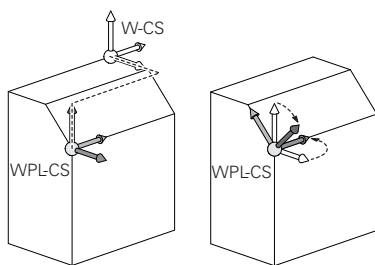
Transformations in the working plane coordinate system WPL-CS

HEIDENHAIN recommends using the following transformations in the working plane coordinate system **WPL-CS**:

■ Axes **X**, **Y**, **Z** of the **TRANS DATUM** function

Further information: "Datum shift with TRANS DATUM", Page 727

■ The **TRANS MIRROR** function or Cycle **8 MIRRORING**

Further information: "Mirroring with TRANS MIRROR", Page 729

Further information: "Cycle 8 MIRRORING", Page 717

■ The **TRANS ROTATION** function or Cycle **10 ROTATION**

Further information: "Rotations with TRANS ROTATION", Page 732

Further information: "Cycle 10 ROTATION ", Page 718

■ The **TRANS SCALE** function or Cycle **11 SCALING FACTOR**

Further information: "Scaling with TRANS SCALE", Page 733

Further information: "Cycle 11 SCALING FACTOR ", Page 720

■ Cycle **26 AXIS-SPECIFIC SCALING**

Further information: "Cycle 26 AXIS-SPECIFIC SCALING ", Page 721

■ The **PLANE RELATIV** function (#8 / #1-01-1)

Further information: "PLANE RELATIV", Page 763

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.
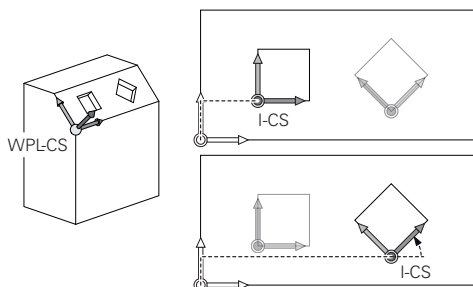
NOTICE

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

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 702

- 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 (#8 / #1-01-1), **PLANE RELATIV** is in 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**.

16.1.7 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 699

If no transformations are defined in the **WPL-CS**, then the position and orientation of the **WPL-CS** and **I-CS** are identical.
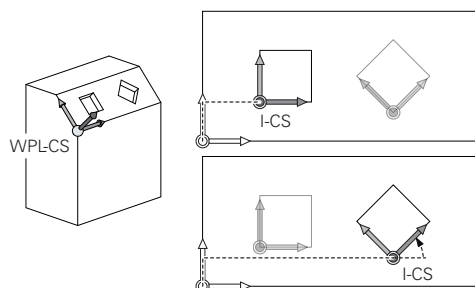
Positioning blocks in the input coordinate system I-CS

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 703

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 (#9 / #4-01-1)
- 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 699

16.1.8 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 196

You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**

Further information: "Tool carrier reference point", Page 253

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 702

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 967

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** (#9 / #4-01-1)

    Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986

- The **FUNCTION TCPM** function (#9 / #4-01-1)

    Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
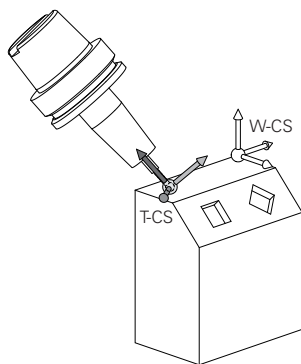
Use the miscellaneous function **M128** to define the tool angle of inclination in the machine coordinate system **M-CS** using axis angles. The effects of the tool angle of inclination depend on the machine kinematics:

Further information: "Notes", Page 989

11 L X+10 Y+45 A+10 C+0 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 TX0 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 tool radius", Page 796
- Compensation values from the tool call

  Further information: "Tool compensation for tool length and tool radius", Page 796
- Values of the compensation tables ***.tco**

  Further information: "Tool compensation with compensation tables", Page 801
- 3D tool compensation with surface normal vectors (#9 / #4-01-1)

  Further information: "3D tool compensation (#9 / #4-01-1)", Page 805

16.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include, for example, 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 196

Use the preset management in the following cases:

- To tilt the working plane of a machine with table or head rotation axes (#8 / #1-01-1)
- 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 *.pr", Page 1710

Description of function

Setting presets

Presets can be set in the following ways:

- Setting axis positions manually
  Further information: "Setting a preset manually", Page 708
- Touch probe cycles in the **Setup** application
  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257
- Touch probe cycles in the NC program (#17 / #1-05-1)
  Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
  Further information: "Cycle 247 PRESETTING ", Page 722

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 1716

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.
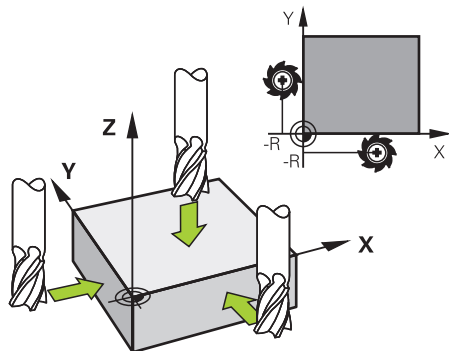
When scratching with a milling cutter, the tool is slowly moved to the workpiece edge in the **Manual operation** application while the spindle is rotating.

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 708

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. If the previous value is kept, there is a danger of collision!

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

Presets can be activated in the following ways:

■ Activating manually in the **Tables** operating mode
Further information: "Activating a preset manually", Page 709

■ Cycle **247 PRESETTING**
Further information: "Cycle 247 PRESETTING ", Page 722

■ **PRESET SELECT** function
Further information: "Activating the preset with PRESET SELECT", Page 710

When activating a preset, the control resets the following transformations:

■ Datum shift with the **TRANS DATUM** function

■ Mirroring 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 697

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.

You can use the **3D ROT** (#8 / #1-01-1) button to define whether the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application.

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

When a basic rotation or 3D basic rotation is active, the control displays a symbol in the **Positions** workspace.

Further information: "Active functions", Page 152

16.2.1   Setting a preset manually
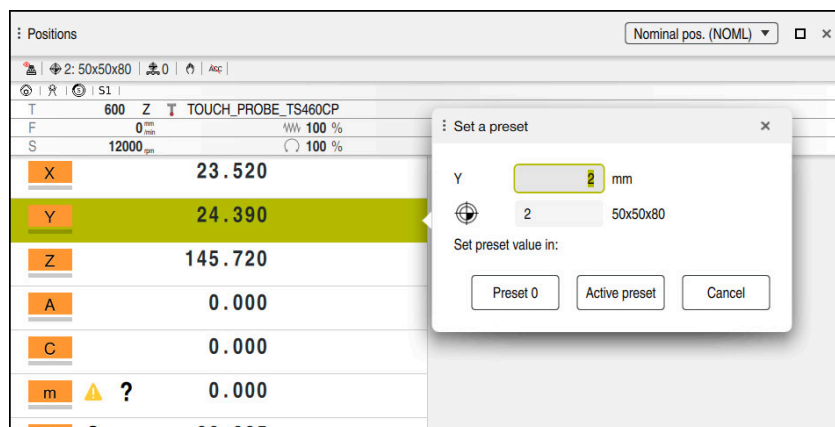
The **Set a preset** window in the **Positions** workspace

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., for 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 for selection.

Active preset
▶ 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.

16.2.2 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. If the previous value is kept, there is a danger of collision!
>
> ▶ Before activating a preset, check whether all columns contain values.
>
> ▶ For undefined columns, enter values (e.g., **0**)
>
> ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

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 displays the number and comment of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 149

Further information: "Status overview on the TNC bar", Page 155

Notes

■ In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every 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 (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.
Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781
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: "The MDI Application ", Page 1221

■ 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 1635

16.3 NC functions for preset management

16.3.1 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

16.3.2 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 705
■ Workpiece preset has been defined
  Further information: "Setting a preset manually", Page 708

Description of function

To activate the preset, use the row number or the content in the **DOC** column.

> ### *NOTICE*
>
> **Danger of collision!**
>
> Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.
>
> ▶ Uniquely define the content of the **DOC** column
>
> ▶ Only activate the preset with the row 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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ PRESET ▶ PRESET SELECT

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET SELECT	Syntax initiator for activating a preset
#, **Name** or **QS**	Select the row of the preset table
	Fixed or variable number or name
	Selection by means of a selection window
	With Name, the control displays in the selection window only the rows of the preset table for which 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

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. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

- If you program **PRESET SELECT** without optional parameters, then the behavior is identical to Cycle **247 PRESETTING**.

  Further information: "Cycle 247 PRESETTING ", Page 722
- If the pallet preset changes, you need to reset the workpiece preset.

  Further information: "Pallet preset table", Page 1635
- With the optional machine parameter **CfgColumnDescription** (no. 105607), the machine manufacturer defines whether the contents of the **DOC** column of the preset table must be unique. If the machine parameter is defined with **TRUE**, you can enter content once only.

16.3.3   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 705

■ Workpiece preset has been defined

Further information: "Setting a preset manually", Page 708

Description of function

To select the preset to be copied, use the row number or the entry in the **DOC** column.

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ PRESET ▶ PRESET COPY

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET COPY	Syntax initiator for copying and activating a workpiece preset
#, **Name** or **QS**	Select the row of the preset table to be copied
	Fixed or variable number or name
	The row can be chosen from a selection menu. With names, the control displays in the selection menu only the rows of the preset table for which the **DOC** column is defined.
TO #, **Name** or **QS**	Select the new row of the preset table
	Fixed or variable number or name
	Selection by means of a selection window
	With Name, the control displays in the selection window only the rows of the preset table for which the **DOC** column is defined.
SELECT TARGET	Activate the copied row of the preset table as the workpiece preset
	Optional syntax element
KEEP TRANS	Retain simple transformations
	Optional syntax element

NOTICE

Danger of collision!

Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.

▶ Uniquely define the content of the **DOC** column
▶ Only activate the preset with the row number

16.3.4 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 705
■ Workpiece preset has been defined

Further information: "Setting a preset manually", Page 708

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 690

Input

11 PRESET CORR X+10 SPC+45	; Correct the workpiece preset in **X** by +10 mm and in **SPC** by +45°

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ PRESET ▶ PRESET CORR

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_OF-FS**, **Z_OFFS**, **A_OFFS**, **B_OF-FS**, **C_OFFS**, **U_OFFS**, **V_OF-FS**, **W_OFFS**	Compensation value for the offsets, referenced to the machine datum Optional syntax element

16.4 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics

- Contents and creation of a datum table
  Further information: "Datum table *.d", Page 1721
- Editing a datum table during a program run
  Further information: "Compensation during program run", Page 1658
- Preset table
  Further information: "Preset table *.pr", Page 1710

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**.

16.4.1 Activating the datum table in the NC program

To activate a datum table in the NC program:

Insert NC function

▶ 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 **Select**

If the datum table is not stored in the same directory as the NC program, the complete path name must be defined. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 206

ⓘ   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

16.5 Coordinate transformation cycles

16.5.1 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 takes effect 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

16.5.2 Cycle 8 MIRRORING

ISO programming

G28

Application

The control can machine the mirror image of a contour in the working plane.

Mirroring takes effect 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**.

Related topics

- Mirroring with **TRANS MIRROR**

  Further information: "Mirroring with TRANS MIRROR", Page 729

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 axes 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
```

16.5.3 Cycle 10 ROTATION

ISO programming
G73

Application
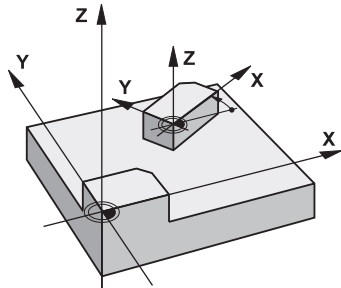
Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle takes effect 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°.

Related topics
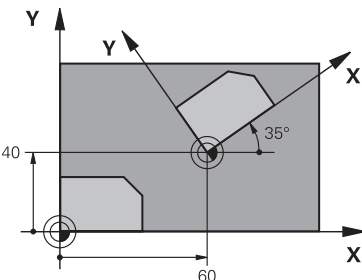
- Rotation with **TRANS ROTATION**
  Further information: "Rotations with TRANS ROTATION", Page 732

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

16.5.4 Cycle 11 SCALING FACTOR

ISO programming
G72

Application
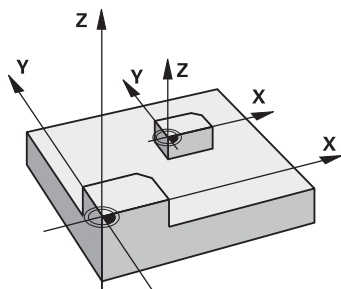
The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The scaling factor takes effect 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.

Related topics

- Scaling with **TRANS SCALE**
  Further information: "Scaling with TRANS SCALE", Page 733

Cycle parameters
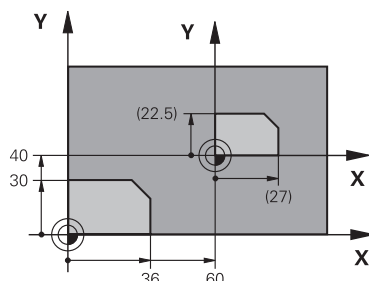
Help graphic	Parameter
	Factor? Enter the scaling factor SCL. The control multiplies the coordinates and radii with SCL. Input: **0.000001...99.999999**

Example

11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75

16.5.5 Cycle 26 AXIS-SPECIFIC SCALING

ISO programming

NC syntax is available only in Klartext programming.
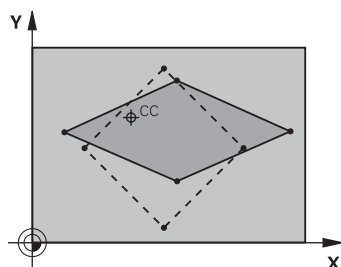
Application

Use Cycle **26** to account for shrinkage and allowance factors for each axis.

The scaling factor takes effect 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.
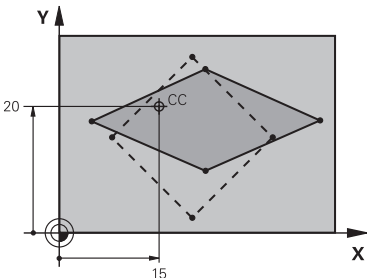
Notes

- 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

16.5.6 Cycle 247 PRESETTING

ISO programming

G247

Application
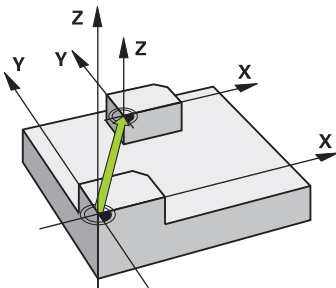
Use Cycle **247 PRESETTING** to activate a preset defined in the preset table as the new preset.

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.

Related topics

- Activate the preset

  Further information: "Activating the preset with PRESET SELECT", Page 710

- Copy the preset

  Further information: "Copying the preset with PRESET COPY", Page 712

- Correct the preset

  Further information: "Correcting the preset with PRESET CORR", Page 714

- Setting and activating presets

  Further information: "Preset management", Page 705

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. If the previous value is kept, there is a danger of collision!
>
> ▶ Before activating a preset, check whether all columns contain values.
>
> ▶ For undefined columns, enter values (e.g., **0**)
>
> ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- 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

16.5.7 Example: Coordinate conversion cycles

Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram
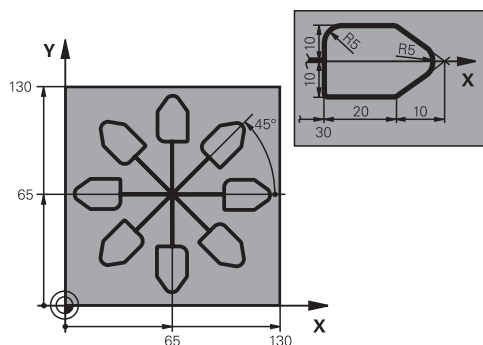
0	BEGIN PGM C220 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	
2	BLK FORM 0.2 X+130 Y+130 Z+0	
3	TOOL CALL 1 Z S4500	; Tool call
4	L Z+100 R0 FMAX M3	; Retract the tool
5	TRANS DATUM AXIS X+65 Y+65	; Shift datum to center
6	CALL LBL 1	; Call milling operation
7	LBL 10	; Set label for program-section repeat
8	CYCL 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 of 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	

16.6 NC functions for coordinate transformation

16.6.1 Overview

The control provides the following **TRANS** functions:

Syntax	Meaning	Further information
TRANS DATUM	Shift the workpiece datum	Page 727
TRANS MIRROR	Mirror an axis	Page 729
TRANS ROTATION	Rotation about the tool axis	Page 732
TRANS SCALE	Scale contours and positions	Page 733
TRANS RESET	Reset the coordinate transformation	Page 734

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 702
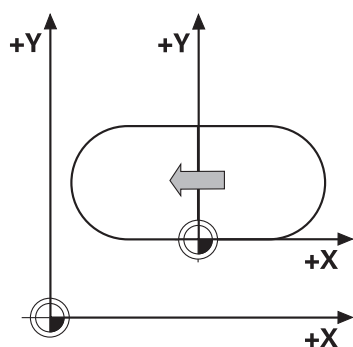
Related topics

- Coordinate transformation cycles

  Further information: "Coordinate transformation cycles", Page 716
- **PLANE** functions (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737
- Reference systems

  Further information: "Reference systems", Page 690

16.6.2 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 *.d", Page 1721
- Activating the datum table

  Further information: "Activating the datum table in the NC program", Page 716
- Machine presets

  Further information: "Presets in the machine", Page 196

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: "The Positions workspace", Page 149

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 716

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 166

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ TRANSFORM ▶ TRANS DATUM

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
Name or **QS**	Path to the datum table Fixed or variable path Selection by means of a selection window 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 an older control, then, during editing, the control turns Cycle **7** into the **TRANS DATUM** NC function.
- If you execute an absolute datum shift with **TRANS DATUM** or Cycle **7 DATUM SHIFT**, then the control overwrites the values of the current datum shift. The control adds the incremental values to the values of the current datum shift.
- Absolute values reference the workpiece preset. Incremental values reference the workpiece datum.
  Further information: "Presets in the machine", Page 196
- A datum shift in the axes **A**, **B**, **C**, **U**, **V** and **W** is effective as an offset. HEIDENHAIN recommends inclining rotary axes using the **PLANE** functions or a 3D basic rotation.
  Further information: "Comparison of offset and 3D basic rotation", Page 1288
- 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 690

16.6.3 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 mirroring.

Related topics

■ Cycle **8 MIRRORING**
  Further information: "Cycle 8 MIRRORING", Page 717

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 196
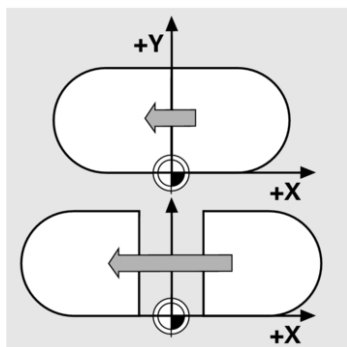
If you mirror only one axis, the machining direction of the tool is reversed. The rotational direction defined in a cycle will remain unchanged (e.g., if defined within one of the OCM cycles (#167 / #1-02-1)).
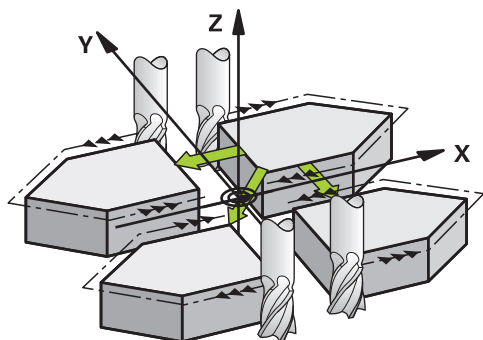
Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **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 of milling machines", Page 194

You can select up to three axis values.
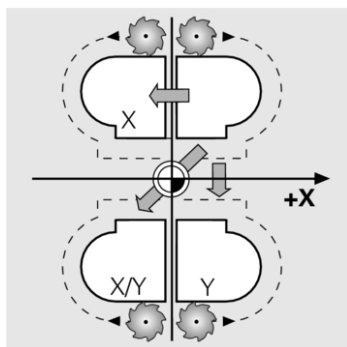
If mirroring is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 166

Input

11 TRANS MIRROR AXIS X	; Mirror X coordinates about the Y 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

Notes

- This function can be used only in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 242

- If you execute mirroring with **TRANS MIRROR** or Cycle **8 MIRRORING**, then the control overwrites the current mirroring.
  Further information: "Cycle 8 MIRRORING", Page 717

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 697

16.6.4 Rotations with TRANS ROTATION

Application

With the **TRANS ROTATION** function, you can rotate contours or positions about a rotation angle.

The **TRANS ROTATION RESET** function allows you to reset the rotation.

Related topics

■ Cycle **10 ROTATION**

Further information: "Cycle 10 ROTATION ", Page 718

Description of function

Rotation is a modal function that is in effect 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 196

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 of milling machines", Page 194
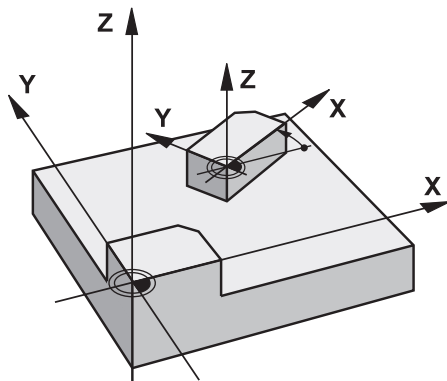
A rotation can be programmed as follows:

■ 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 166

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

Notes

■ This function can be used only in the **FUNCTION MODE MILL** machining mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 242

■ If you execute an absolute rotation with **TRANS ROTATION** or Cycle **10 ROTATION**, then the control overwrites the values of the current rotation. The control adds the incremental values to the values of the current rotation.

Further information: "Cycle 10 ROTATION ", Page 718

16.6.5 Scaling with TRANS SCALE

Application

The **TRANS SCALE** function lets you change the scale of the contours or distances to the datum, thereby evenly enlarging or shrinking them. This enables you to program shrinkage and oversize allowances, for example.

Use the **TRANS SCALE RESET** function to reset scaling.

Related topics

■ Cycle **11 SCALING FACTOR**

Further information: "Cycle 11 SCALING FACTOR ", Page 720

Description of function

Scaling is a modal function that is in effect 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 196
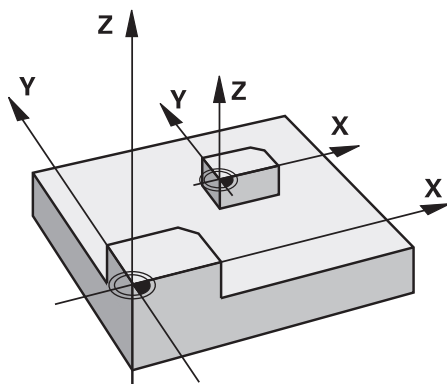
If you enter a scaling factor **SCL** less than 1, the contour will be reduced in size. If you enter a scaling factor **SCL** greater than 1, the contour will be enlarged.

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 166

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 be used only in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 242
- If you execute a change of scale with **TRANS SCALE** or Cycle **11 SCALING FACTOR**, then the control overwrites the current scaling factor.
  Further information: "Cycle 11 SCALING FACTOR ", Page 720
- 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.

16.6.6 Resetting with TRANS RESET

Application

Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.

Related topics

- NC functions for coordinate transformation
  Further information: "NC-Funktionen zur Koordinatentransformation", Page
- Coordinate transformation cycles
  Further information: "Coordinate transformation cycles", Page 716

Description of function

The control resets the following simple coordinate transformations:

Coordinate transformation	Syntax	Further information
Datum shift	**TRANS DATUM**	Page 727
Mirroring	**TRANS MIRROR**	Page 729
	Cycle **8 MIRRORING**	Page 717
Rotation	**TRANS ROTATION**	Page 732
	Cycle **10 ROTATION**	Page 718
Scaling	**TRANS SCALE**	Page 733
	Cycle **11 SCALING FACTOR**	Page 720
	Cycle **26 AXIS-SPECIFIC SCALING**	Page 721

> **ⓘ** The control also resets simple coordinate transformations defined by the machine manufacturer.

Input

11 TRANS RESET	; Reset simple coordinate transformations

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **TRANSFORM** ▶ **TRANS RESET**

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS RESET	Syntax opener for resetting simple coordinate transformations

16.7 Tilting the working plane (#8 / #1-01-1)

16.7.1 Fundamentals

Machines with rotary axes allow machining of, for example, 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 699
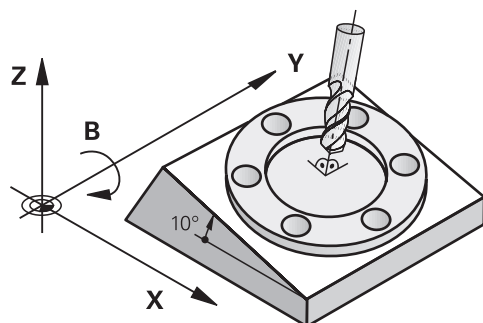
Two functions are available for tilting the working plane:

■ Manual tilting with the **3-D rotation** window in the **Manual operation** application

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

■ Tilting under program control with the **PLANE** functions in the NC program

Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737

> ℹ️ You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

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 695

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 kinematic model, 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 699
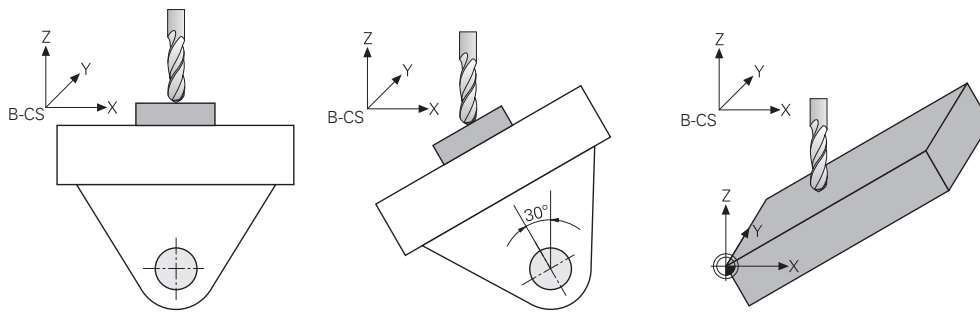
- Machine with head rotary axes

  With this kinematic model, 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 699
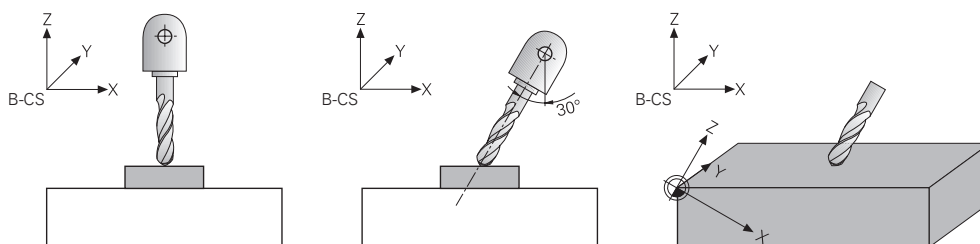
16.7.2 Tilting the working plane with PLANE functions (#8 / #1-01-1)

Fundamentals

Application

Machines with rotary axes allow machining of, for example, 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 950
- Adopting a tilted working plane in the **Manual** operating mode with the **3-D rotation** window
  Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

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.
- Software option Advanced Functions Set 1 (#8 / #1-01-1)
- 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 690

> 🛈 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 in effect 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 727

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 739

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 771

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 774

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 778

Status display

The Positions workspace

As soon as the working plane has tilted, the General status display in the **Positions** workspace contains an icon.

Further information: "The Positions workspace", Page 149

> ℹ When deactivating or resetting the tilting function correctly, the icon indicating the tilted working plane must disappear.
> **Further information:** "PLANE RESET", Page 767

The 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 164

Further information: "TRANS tab", Page 166

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 742
PROJECTED	Defines the working plane by means of two projection angles and one rotation angle	Page 748
EULER	Defines the working plane by means of three Euler angles	Page 752
VECTOR	Defines the working plane by means of two vectors	Page 755
POINTS	Defines the working plane by means of the coordinates of three points	Page 758
RELATIV	Defines the working plane by means of a single spatial angle with incremental effect	Page 763
AXIAL	Defines the working plane by means of a maximum of three absolute or incremental axis angles	Page 768
RESET	Resets tilting of the working plane	Page 767

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 reset all **PLANE** functions with **PLANE RESET**. For example, if you define all spatial angles with 0, the control resets only the angles and not the tilting function.

▪ If you restrict the number of rotary axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine manufacturer decides whether the control takes the angles of deselected axes into account or sets them to 0.

■ The control only supports tilting functions if tool axis **Z** is active.

■ If necessary, you can edit Cycle **19 WORKING PLANE**. However, you cannot insert the cycle again, because the control no longer offers the cycle for programming.

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

ℹ 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 763
- Entering the axis angle
  Further information: "PLANE AXIAL", Page 768

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.
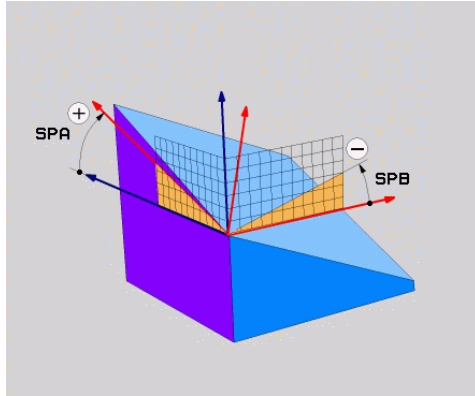
Spatial angles **SPA** and **SPB**
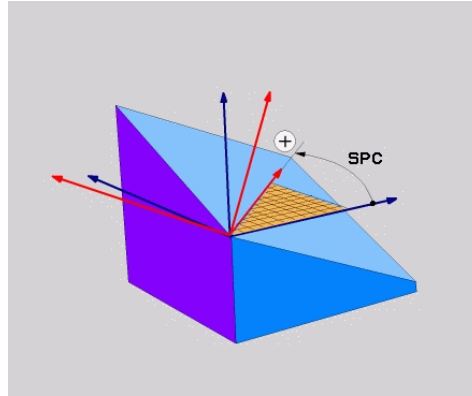
Spatial angle **SPC**

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.
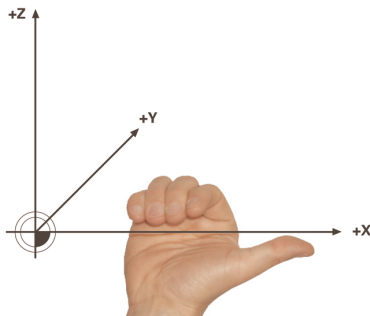
The thumb of your right hand points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

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 746

> 🛈 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 763

Application example

Example

11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
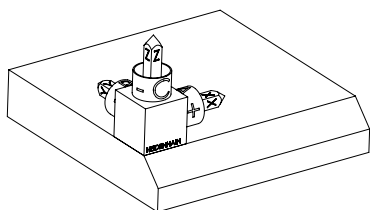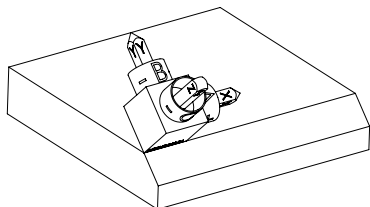
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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 746
- **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	Rotation around the X axis of the workpiece coordinate system **W-CS** Input: **-360.0000000…+360.0000000**
SPB	Rotation around the Y axis of the **W-CS** Input: **-360.0000000…+360.0000000**
SPC	Rotation around the Z axis of the **W-CS** Input: **-360.0000000…+360.0000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning ℹ 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 771
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 774 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 778 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
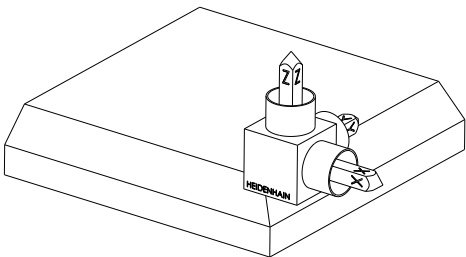
Initial state
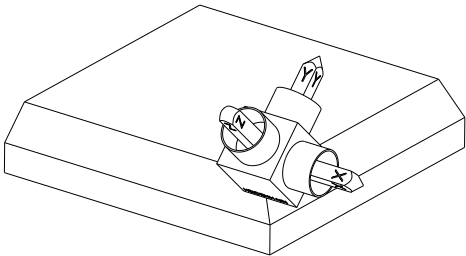
SPA+45

Orientation of tool axis **Z**

Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**
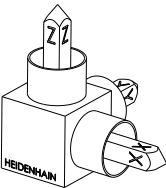
SPB+0

Rotation around the Y axis of the non-tilted **W-CS**

No rotation with value 0
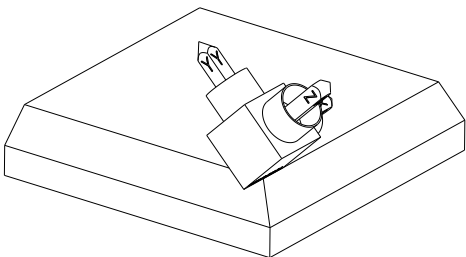
SPC+90

Orientation of main axis **X**

Rotation around the Z axis of the non-tilted **W-CS**
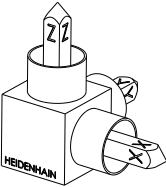
View C-B-A

Initial state
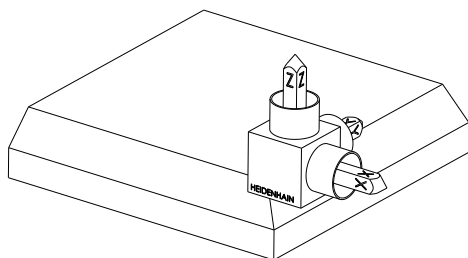
SPC+90

Orientation of main axis **X**

Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane
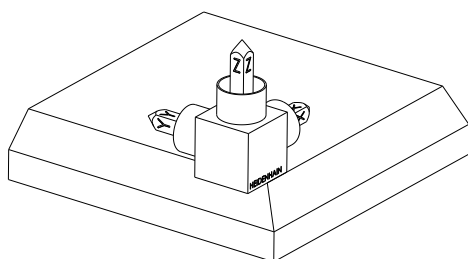
SPB+0

Rotation around the Y axis in the working plane coordinate system **WPL-CS**, meaning in the tilted working plane

No rotation with value 0
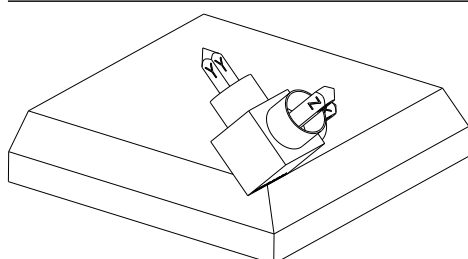
SPA+45

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 of milling machines", Page 194

Use an additional rotation angle to optionally align the X axis in the tilted working plane.
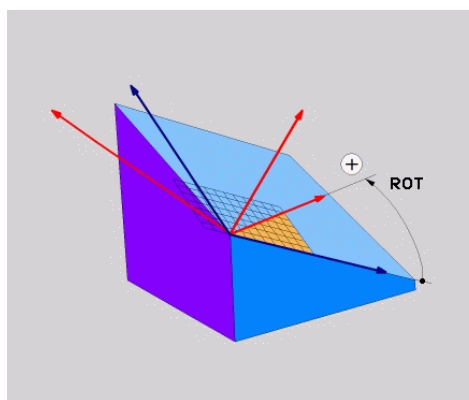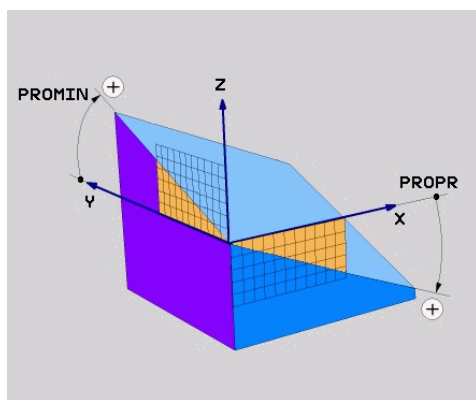
Projection angles **PROMIN** and **PROPR**   Rotation angle **ROT**

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 751

Application example

Example

> **11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE ROT**
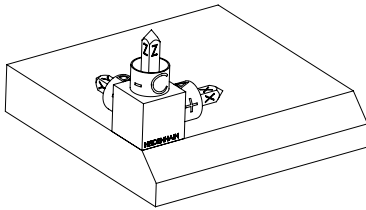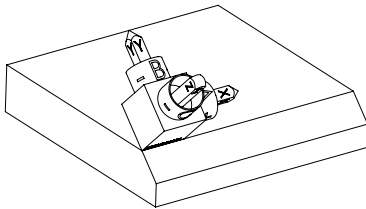
Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around 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 active 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 ℹ️ 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 771
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 774 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 778 Optional syntax element

Notes

Procedure in case of hidden workpiece edges, using the example of a diagonal hole
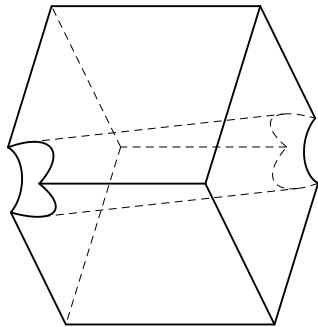
Cube with a diagonal hole
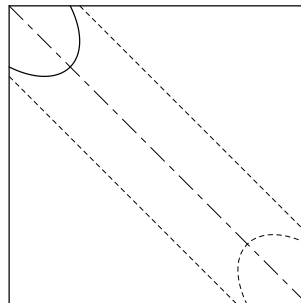
Front view, meaning projection on the **ZX** working plane

Example

> 11 PLANE PROJECTED PROPR-45 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM-
> TABLE ROT

Comparison of projection and spatial angles

45°
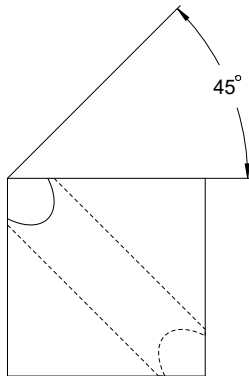
When imagining the workpiece to be transparent, the projection angles are easy to find.

Both projection angles are 45°.

> **i** When defining the algebraic sign, ensure that the working plane is perpendicular to the center axis of the hole.

54.736°
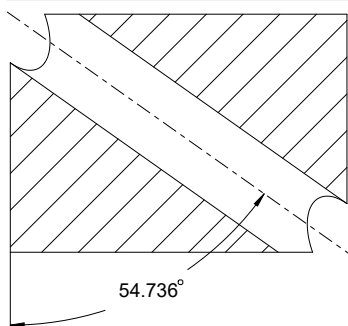
When defining the working plane by using spatial angles, the spatial diagonal must be considered.

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.
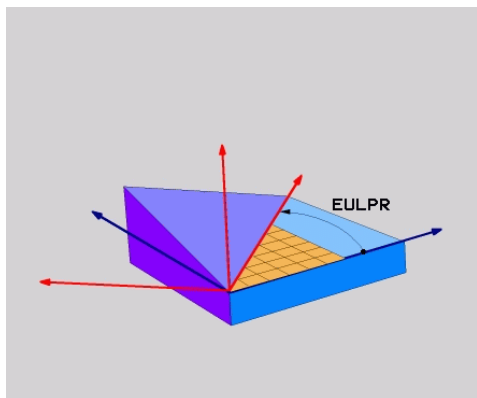
Euler angle **EULPR**
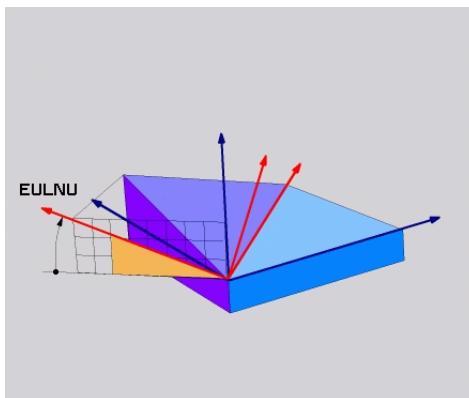
Euler angle **EULNU**
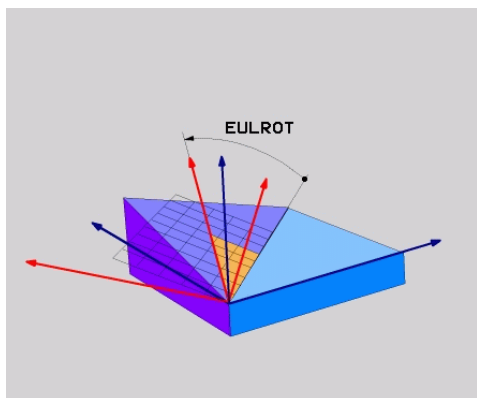
Euler angle **EULROT**

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.

> ℹ️ 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 763
>
> 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 742
>
> **Further information:** "Rotations with TRANS ROTATION", Page 732

Application example

Example

11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
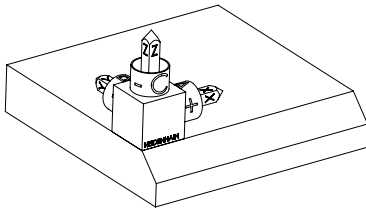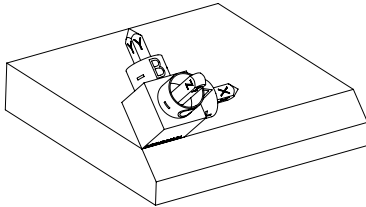
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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.

i 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 771
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 774 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 778 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 948

Description of function

Vectors define a working plane as two independent specifications of direction, starting from the non-tilted workpiece coordinate system **W-CS**.
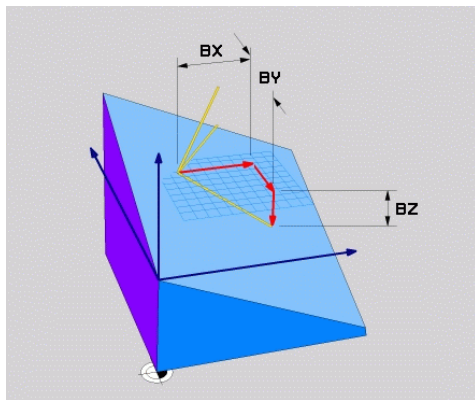
Base vector with components **BX**, **BY** and **BZ**
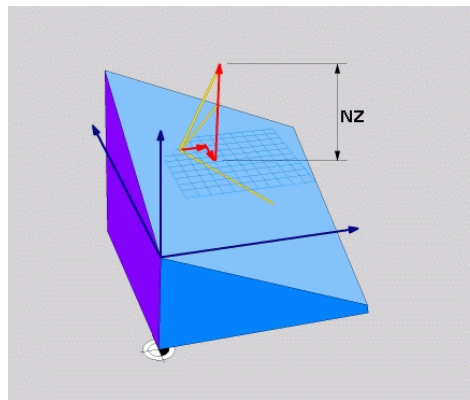
NZ component of the normalized vector

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 756

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
```
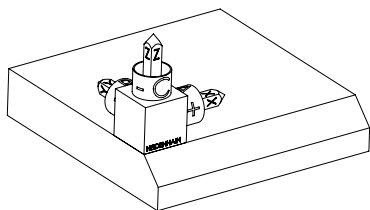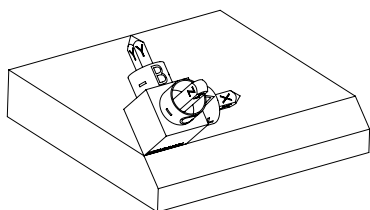
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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 ℹ 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 771
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 774 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 778 Optional syntax element

Notes

- If the components of the normal vector contain very small values, such as 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 (#17 / #1-05-1)", Page 1527

Description of function

Points define a working plane by using their coordinates in the non-tilted workpiece coordinate system **W-CS**.
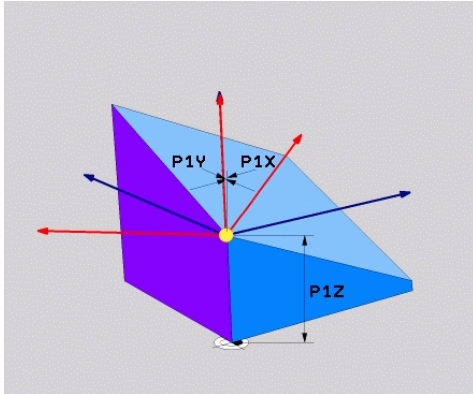
First point with coordinates **P1X**, **P1Y** and **P1Z**
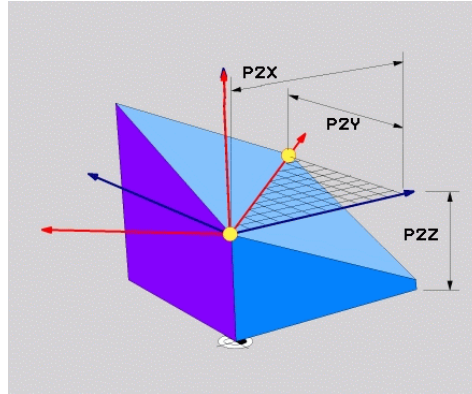
Second point with coordinates **P2X**, **P2Y** and **P2Z**
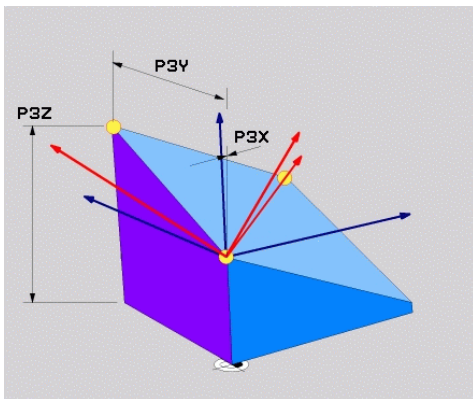
Third point with coordinates **P3X**, **P3Y** and **P3Z**

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.

> ℹ️ 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.

> ℹ️ 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.
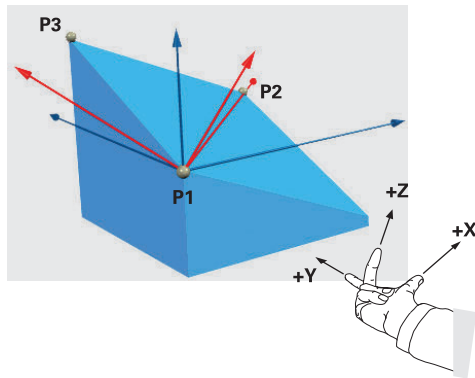
To direct the positive tool axis direction away from the workpiece, the following conditions apply to the position of the three points:

- 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
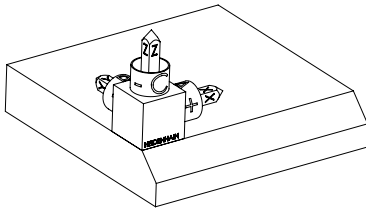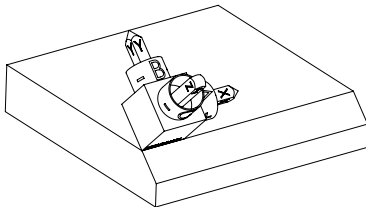
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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.

> **i** 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**.

> **i** 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 771
SYM or **SEQ**	Select an unambiguous tilting solution
	Further information: "Tilting solution", Page 774
	Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type
	Further information: "Transformation types", Page 778
	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 690

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**.

> ℹ️ **PLANE RELATIV** allows, for example, programming a chamfer on a tilted workpiece surface by tilting the working plane further by the chamfer angle.
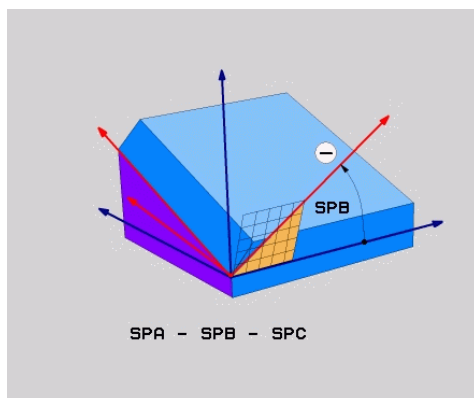
Additive spatial angle **SPB**

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**
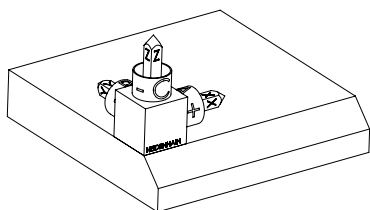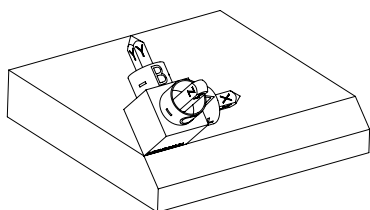
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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 766

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 in effect 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 771
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 774 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 778 Optional syntax element

Note

Incremental datum shift using a chamfer as example
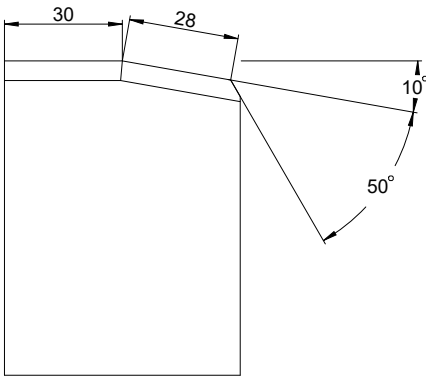
50° chamfer on a tilted workpiece
surface

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

  The function does not reset any offset values!

  Further information: "Basic transformation and offset", Page 1714
- 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 771

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 771

Notes

- Before every program run, ensure that no undesired coordinate transformations are in effect. When needed, tilting of the working plane can also be deactivated manually in the **3-D rotation** window.

  Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

  > ⓘ The status display allows checking the desired status of the tilting situation.
  >
  > **Further information:** "Status display", Page 739

- With the touch probe functions you can save the misalignment of the workpiece as a 3D basic rotation in the preset table (e.g., **Plane (PL)**). In the NC program you must then align the workpiece with a tilting function (e.g., with **PLANE SPATIAL SPA+0 SPB+0 SPC+0 TURN FMAX**). You must not use **PLANE RESET** for machining, because the control does not take into account the 3D basic rotation with this function.

  Further information: "PLANE SPATIAL", Page 742

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.

> **i** Because you are able to define just one axis angle, you can also use **PLANE AXIAL** on machines with just 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 742

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.
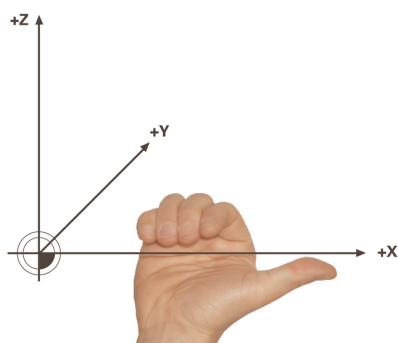
Extended right-hand rule for head rotary axes
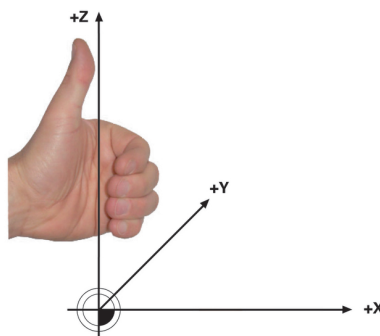
Extended left-hand rule for table rotary axes

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**.
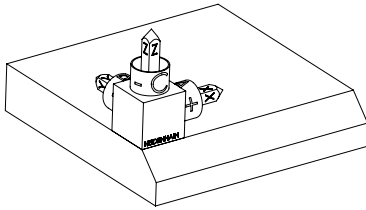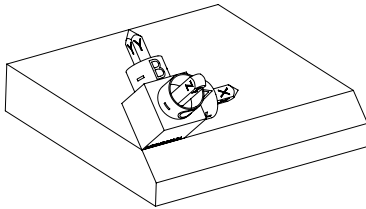
Orientation of the tool axis	Using the defined axis angle **A**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by angle **A** is around the non-tilted X axis.

> **i** 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.
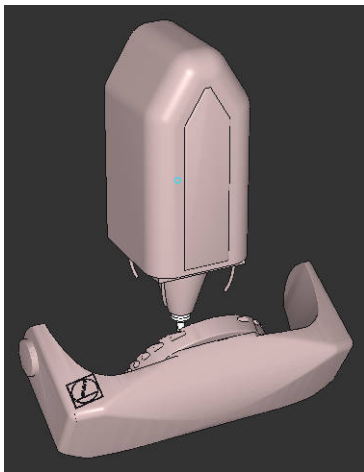
The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> **i** 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

> ℹ️ 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 771

> ℹ️ 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 698
- 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 in part 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.
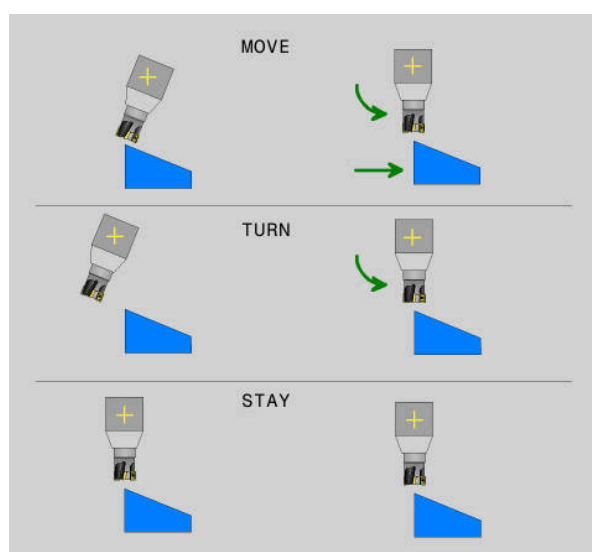
Type of rotary axis positioning	Meaning
MOVE	If you perform tilting near the workpiece, then use this option.
	Further information: "Rotary axis positioning with MOVE", Page 772
TURN	If the workpiece is so large that the range of traverse is not sufficient for the compensating movement of the linear axes, then use this option.
	Further information: "Rotary axis positioning TURN", Page 772
STAY	The control does not position any axes.
	Further information: "Rotary axis positioning with STAY", Page 773

Rotary axis positioning with MOVE

The control positions the rotary axes and performs compensation movements in the linear main axes.

The compensation movements ensure that the relative position between the tool and the workpiece will not change during the positioning process.

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. Please note that the compensation movement is performed in up to three axes.

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 you define the value 0, the center of rotation and consequently the center of the compensation movements is in the tool tip.

When you define **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.

🛈 If you wish to tilt about 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.
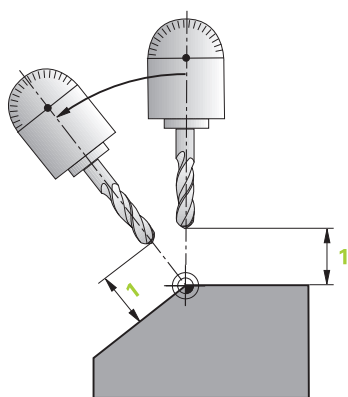
Rotary axis positioning TURN

The control positions only the rotary axes. The tool must be positioned after tilting to position.

Rotary axis positioning with STAY

Both the rotary axes and the tool must be positioned after tilting to position.

> **ℹ** 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 STAY
```

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 775
SEQ	With **SEQ** you select a tilting solution relative to the basic position of the master axis.
	Further information: "Tilting solution SEQ", Page 776
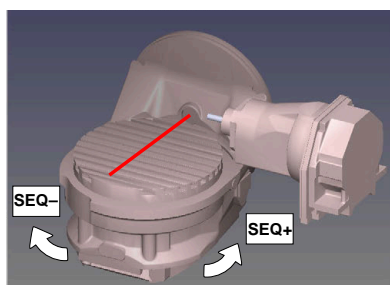
Reference for **SEQ**
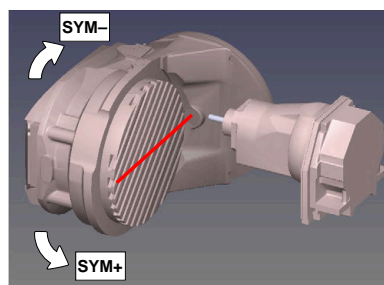
Reference for **SYM**

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
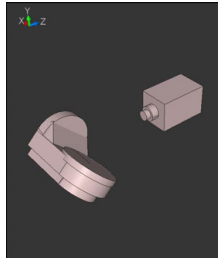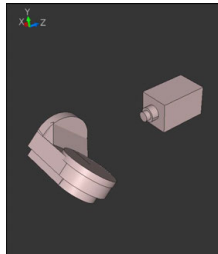
Machine with C rotary axis and A tilting table.
Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

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	

> ℹ️ 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.

> ℹ️ 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
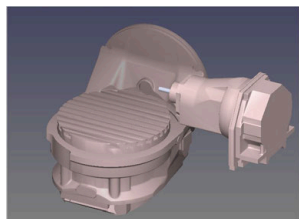
Origin
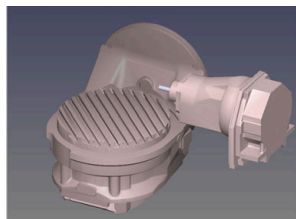
A = 0, B = 45
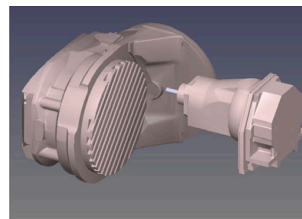
A = −90, B = 45

> 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 is also dependent on a programmed rotation (e.g., with Cycle **10 ROTATION**).

16.7.3 The 3-D rotation window (#8 / #1-01-1)

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 (#8 / #1-01-1)", Page 737
- Reference systems of the control

  Further information: "Reference systems", Page 690

Requirements

- Machine with rotary axes
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Software option Advanced Functions Set 1 (#8 / #1-01-1)
- 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: "The Manual operation application", Page 186
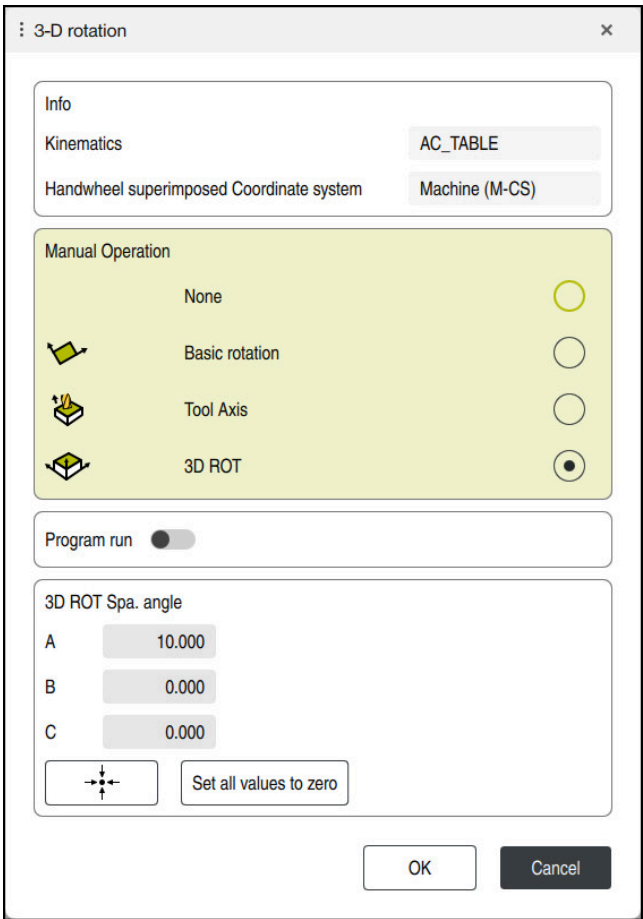
The **3-D rotation** window

The **3-D rotation** window contains the following information:

Area	Contents
Info	Information about the machine:
	■ Name of the active machine kinematics
	■ Coordinate system in which handwheel superimpositioning is active
	Further information: "Reference systems", Page 690
	Further information: "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979
Manual Operation	Effect of the 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 take place in the **W-CS** workpiece coordinate system.
	Further information: "Workpiece coordinate system W-CS", Page 697
	■ **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 take place in the **W-CS** workpiece coordinate system.

Area	Contents
	Further information: "Selection item Basic rotation", Page 783
	■ **Tool axis**
	This is relevant only for head rotary axes. The traverses take place in the **T-CS** tool coordinate system.
	Further information: "The Tool axis selection item", Page 783
	■ **3D ROT**
	The control takes the positions of rotary axes and columns **SPA**, **SPB** and **SPC** of the preset table into account. The traverses take place in the **WPL-CS** working plane coordinate system.
	Further information: "The 3D ROT selection item", Page 784
Program run	When activating the **Tilt working plane** function for the **Program run** operating mode, the entered angle of rotation applies starting 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, then the angular values defined there become active. The control will reset the entered angular values to 0.
3D ROT Spa. angle	Currently active angle for the **3D ROT** selection item
	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.

Confirm the selection with **OK**. If a selection item is active in the **Manual Operation** or **Program run** areas, then the control highlights the area in green.

If a selection item is active in the **3-D rotation** window, then the control displays the appropriate symbol in the **Positions** workspace.

Further information: "The Positions workspace", Page 149

Selection item Basic rotation

If you select **Basic rotation**, then the axes move, taking into account a basic rotation or a 3D basic rotation.

Further information: "Basic rotation and 3D basic rotation", Page 707

The axis movements take effect in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 697

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: "The Positions workspace", Page 149

With this selection item, the **3D ROT Spa. angle** area has no function.

The Tool axis selection item

If you select **Tool axis**, then you can move in the positive or negative direction of the tool axis. The control locks all other axes. This selection item makes sense only for machines with rotary head axes.

The traverse movement is active in the **T-CS** tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 703

This selection item can be used, for example, in the following cases:

- 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.

With this selection item, the **3D ROT Spa. angle** area has no function.

The 3D ROT selection item

If you select **3D ROT**, then all axes move in the tilted machining plane. The traversing movements are active in the **WPL-CS** working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 699

If a basic rotation or 3D basic rotation has additionally been saved to the preset table, then it will automatically be taken into account.

In the **3D ROT Spa. angle** area, the control shows the currently active angle. The spatial angle can also be edited.

> If you edit the values in the **3D ROT Spa. angle** area, then you must position the rotary axes (e.g., in the **MDI** application).

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

  > **COORD ROT** is only possible with a free rotary axis.
  > **Further information:** "Transformation types", Page 778

- 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 (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

  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: "The Referencing workspace", Page 181

- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.

16.8 Inclined machining (#9 / #4-01-1)

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** (#9 / #4-01-1)
  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
- Compensating the tool angle of inclination with **M128** (#9 / #4-01-1)
  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986
- Tilting the working plane (#8 / #1-01-1)
  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736
- Presets on the tool
  Further information: "Presets on the tool", Page 253
- Reference systems
  Further information: "Reference systems", Page 690

Requirements

- Machine with rotary axes
- Kinematics description
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Software option Advanced Functions Set 2 (#9 / #4-01-1)

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 (#8 / #1-01-1)", Page 736

Inclined machining can be implemented using the following functions:

- Incremental traverse of rotary axis
  Further information: "Inclined machining with incremental process", Page 786
- Normal vectors
  Further information: "Inclined machining using normal vectors", Page 786

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
* - ...	

16.9   Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)

Application

The **FUNCTION TCPM** function allows you to influence the positioning behavior of the control. When activating **FUNCTION TCPM**, the control compensates for any changed tool angles of inclination by means of compensating movements of the linear axes.

FUNCTION TCPM allows, for example, changing the tool angle of inclination for inclined machining while the position of the tool location point relative to the contour remains the same.

> **i**   Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

- Compensating for the tool angle of inclination with **M128**

    Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986

- Tilting the working plane

    Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736

- Presets on the tool

    Further information: "Presets on the tool", Page 253

- Reference systems

    Further information: "Reference systems", Page 690

Requirements

- Machine with rotary axes
- Kinematics description

    To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.

- Software option Advanced Functions Set 2 (#9 / #4-01-1)

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.
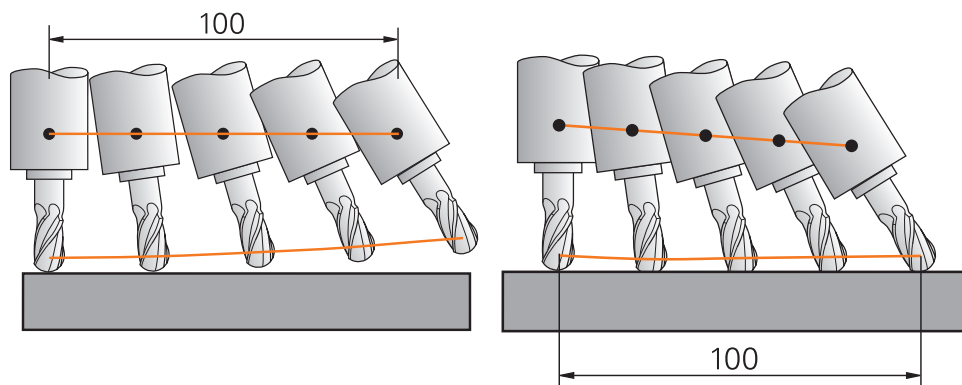
Behavior without **TCPM**

Behavior with **TCPM**

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. Please note that the compensation movement is performed in up to three axes.

When **FUNCTION TCPM** is active, the control shows the **TCPM** icon in the position display.

Further information: "The Positions workspace", Page 149

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 789
AXIS POS or **AXIS SPAT**	Interpretation of programmed rotary axis coordinates **Further information:** "Interpretation of the programmed rotary axis coordinates", Page 790
PATHC-TRL AXIS or **PATHCTRL VECTOR**	Interpolation of tool angle of inclination **Further information:** "Interpolation of tool angle of inclination between start and end positions", Page 791
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 792 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 793 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**
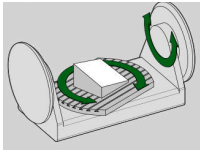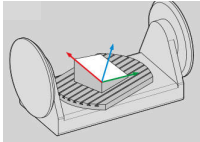
Interpretation of the programmed feed rate

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.
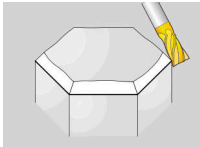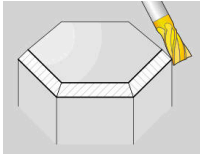
Interpretation of the programmed rotary axis coordinates

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. The **AXIS SPAT** selection item defines the spatial angles relative to the **I-CS** input coordinate system. The defined angles have the effect of incremental spatial angles. In the first traversing block after the function **FUNCTION TCPM**, always program with **AXIS SPAT**, **SPA**, **SPB** and **SPC**, including with spatial angles of 0°. **Further information:** "Input coordinate system I-CS", Page 702

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 902
	PATHCTRL AXIS can be used both for face milling and also for peripheral milling.
	Further information: "3D tool compensation during face milling (#9 / #4-01-1)", Page 809
	Further information: "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 816
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 generates 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.

> ℹ To obtain continuous movement, define Cycle **32** with a **tolerance for rotary axes**.
>
> **Further information:** "Cycle 32 TOLERANCE ", Page 902

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.
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 798
>
> If you use **REFPNT CENTER-CENTER** to program pocket milling cycles, the control generates an error message.
>
> **Further information:** "Milling pockets ", Page 539

Further information: "Presets on the tool", Page 253

The reference point is optional. If you do not enter anything, the control uses **REFPNT TIP-TIP**.
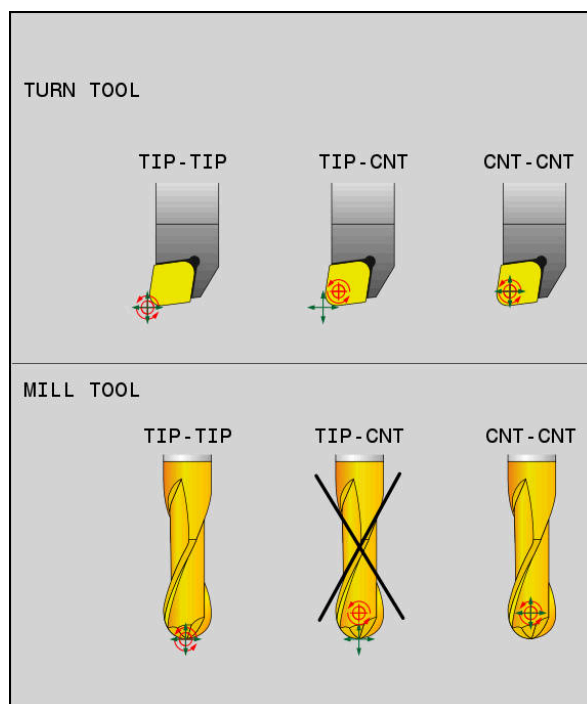
Selection options of tool location point and tool rotation point

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).

> **i** 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 affect only 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.
>
> ▶ Make sure to retract the tool before changing the position of the rotary 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 **444 PROBING IN 3-D** (#17 / #1-05-1)
- **M128** and **FUNCTION TCPM** with **AXIS POS** selected do not take into account an active 3D basic rotation. Program **FUNCTION TCPM** with **AXIS SPAT** selected, or CAM outputs with **LN** straight lines and a tool vector.

  Further information: "Basic rotation and 3D basic rotation", Page 707

  Further information: "Straight line LN", Page 806
- Use only ball-nose cutters for face milling in order to avoid contour damage. In combination with other tool shapes, check the NC program for any possible contour damage by using the **Simulation** workspace.

  Further information: "Notes", Page 989

Notes about machine parameters

The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control will interpret offset values. With **FUNCTION TCPM** and **M128**, the machine parameter is relevant only for the rotary axis that rotates about the tool axis (mostly **C_OFFS**).

Further information: "Basic transformation and offset", Page 1714

- If the machine parameter is not defined or is defined with the value **TRUE**, then you can compensate for a workpiece misalignment in the plane with the offset. The offset affects the orientation of the workpiece coordinate system **W-CS**.

  Further information: "Workpiece coordinate system W-CS", Page 697

- If the machine parameter is defined with the value **FALSE**, then you cannot compensate for a workpiece misalignment in the plane. The control does not take the offset into account during program run.

17

Compensations

17.1   Tool compensation for tool length and tool 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 active in the tool axis. The tool radius delta value **DR** is active exclusively for radius-compensated traverses with the path functions and cycles.

Further information: "Path Functions", Page 291

Related topics

- Tool radius compensation
  Further information: "Tool radius compensation", Page 798
- Tool compensation with compensation tables
  Further information: "Tool compensation with compensation tables", Page 801

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, for example, by using a tool touch probe. The control automatically enters the delta values in the tool management.

  Further information: "Tool management ", Page 269

- Delta values within a tool call serve for a tool compensation that is active exclusively in the current NC program (e.g., a workpiece oversize).

  Further information: "Tool call by TOOL CALL", Page 279
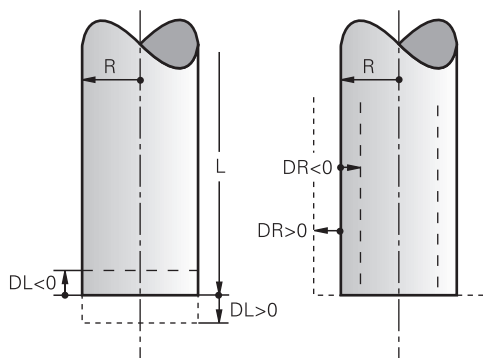
Delta values represent deviations from the length and radius of a tool.

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 279

Further information: "Tool compensation with compensation tables", Page 801

Delta values within a tool call can also be defined by using variables.

Further information: "Tool data within variables", Page 798

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 = $L + DL_{TAB} + DL_{Prog}$

L:	Tool length **L** from the tool table
	Further information: "Tool table tool.t", Page 1684
DL $_{TAB}$:	Tool length delta value **DL** from the tool table
	Further information: "Tool table tool.t", Page 1684
DL $_{Prog}$:	Tool length delta value **DL** from the tool call or the compensation table
	The most recently programmed value becomes active.
	Further information: "Tool call by TOOL CALL", Page 279
	Further information: "Tool compensation with compensation tables", Page 801

NOTICE

Danger of collision!

The control uses the defined tool length from the tool table for compensating for 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 a **TOOL CALL 0**. There is a risk 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 798
- Within machining cycles
  Further information: "Working with cycles", Page 222
- For straight lines **LN** with surface normal vectors
  Further information: "Straight line LN", Page 806

In tool radius compensation, the control takes the delta values from the tool table and the NC program into account.

Active tool radius = $R + DR_{TAB} + DR_{Prog}$

R:	Tool radius **R** from the tool table
	Further information: "Tool table tool.t", Page 1684
DR $_{TAB}$:	Tool radius delta value **DR** from the tool table
DR $_{Prog}$:	Tool radius delta value **DR** from the tool call or the compensation table
	The most recently programmed value becomes active.
	Further information: "Tool call by TOOL CALL", Page 279
	Further information: "Tool compensation with compensation tables", Page 801

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 1013

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

You can use the Q parameter **Q108 ACTIVE TOOL RADIUS** in order to shift the tool center point of the ball-nose cutter to the sphere center using the delta value for the tool length.

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 changes only the position of the tool in the simulation.

    Further information: "Simulation of tools", Page 1207

- 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 call", Page 279

    Further information: "The Positions workspace", Page 149

17.2 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 253

Requirements

- Defined tool data in the tool management
  Further information: "Tool management ", Page 269

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 = $R + DR_{TAB} + DR_{Prog}$

Further information: "Tool compensation for tool length and tool radius", Page 796

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
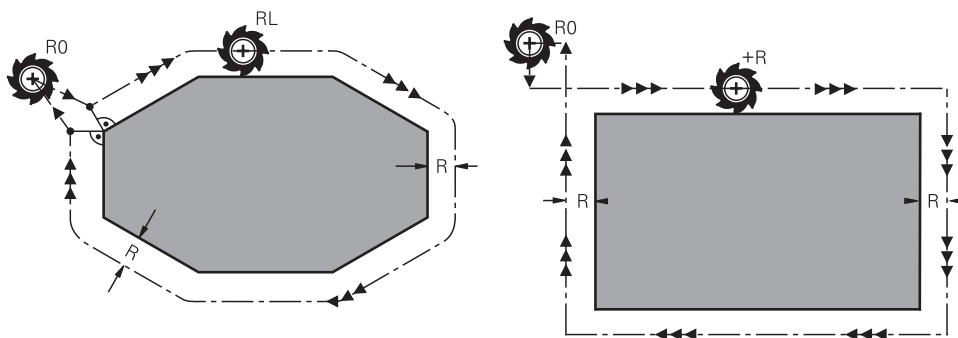
Radius-compensated traverse with path functions

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.
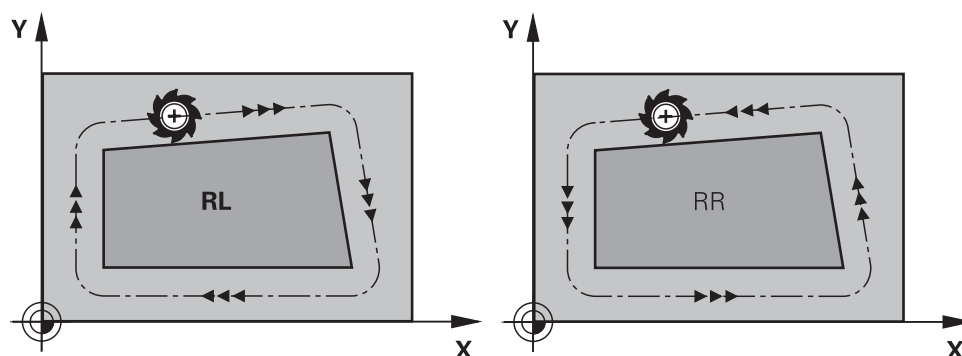
RL: The tool moves on the left of the contour

RR: The tool moves on the right of the contour

Effect

Tool radius compensation is active 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 for quicker implementation of changes, for example.

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 symbol in the **Positions** workspace.
  Further information: "The Positions workspace", Page 149
- Between two NC blocks, each with a different tool radius compensation **RR** and **RL**, there must be at least one traversing block in the working plane without tool radius compensation **R0**.
- If radius compensation is active and you execute the following functions, the control aborts program run and displays an error message:
  - **PLANE** functions (#8 / #1-01-1)
  - **M128** (#9 / #4-01-1)
  - **FUNCTION TCPM** (#9 / #4-01-1)
  - **CALL PGM**
  - Cycle **12 PGM CALL**
  - Cycle **32 TOLERANCE**
  - Cycle **19 WORKING PLANE**

> **ⓘ** You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

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 during, for example, large changes in direction

■ Inside corners:
The control calculates the intersection of the tool center paths at inside corners under radius compensation. Starting at this point, the tool moves along the next contour element. This prevents damage to the workpiece at the inside corners. As a result, the tool radius for a certain contour cannot be selected to be just any size.

17.3 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). You can call the saved compensations during the NC program, in order to compensate for tool values.

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 690

Related topics

■ Contents of the compensation tables
Further information: "Compensation table *.tco", Page 1732
Further information: "Compensation table *.wco", Page 1734

■ Editing compensation tables during program run
Further information: "Compensation during program run", Page 1658

Description of function

In order to compensate tools by using the compensation tables, the steps below are needed:

- Creating a compensation table

  Further information: "The Create new table window", Page 1668
- Activating the compensation table in the NC program

  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 803
- As an alternative, activating the compensation table manually for the program run

  Further information: "Activating the compensation tables manually", Page 802
- Activating a compensation value

  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 804

The compensation table values can be edited within the NC program.

Further information: "Accessing table values ", Page 1679

The values in the compensation tables can be edited even while the program is running.

Further information: "Compensation during program run", Page 1658

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 703

The compensation table **.tco** is the alternative to compensating with **DL**, **DR** and **DR2** in the Tool Call block. As soon as you have activated a compensation table, the control overwrites the compensation value from the Tool Call block.

Further information: "Tool call by TOOL CALL", Page 279

If a shift with the ***.tco** compensation table is active, the control displays it on the **Tool** tab of the **Status** workspace.

Further information: "The Tool tab", Page 168

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 699

If a shift with the ***.wco** compensation table is active, the control displays it, including the path, on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 166

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**.

17.3.1 Selecting a compensation table with SEL CORR-TABLE

Application

If you are using compensation tables, then use the **SEL CORR-TABLE** function 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 804

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 1732

  Further information: "Compensation table *.wco", Page 1734

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**

To navigate to this function:

Insert NC function ▶ All functions ▶ Selection ▶ SEL CORR-TABLE

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**
Name or **QS**	Path of table Fixed or variable name Selection by means of a selection window

17.3.2 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 803

- Contents of the compensation tables

 Further information: "Compensation table *.tco", Page 1732

 Further information: "Compensation table *.wco", Page 1734

Description of function

The activated compensation values are active 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** |

To navigate to this function:

Insert NC function ▶ All functions ▶ Selection ▶ FUNCTION CORRDATA

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
#, **Name** 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

17.4   3D tool compensation (#9 / #4-01-1)

17.4.1   Fundamentals

The control allows 3D tool compensation in CAM-generated NC programs with surface-normal vectors.

Further information: "Straight line LN", Page 806

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 808

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 as a finishing allowance, for example

The situations below are some of the cases where 3D tool compensation can be used:
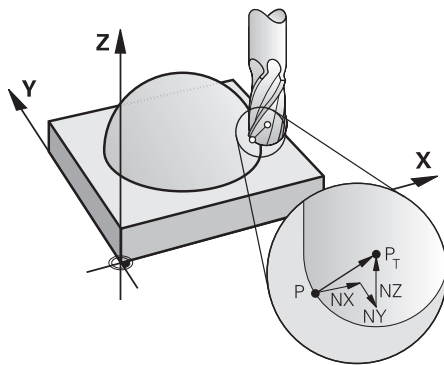
> **i**   For an optional tool angle of inclination, the NC blocks must include an additional tool vector with the components TX, TY and TZ.
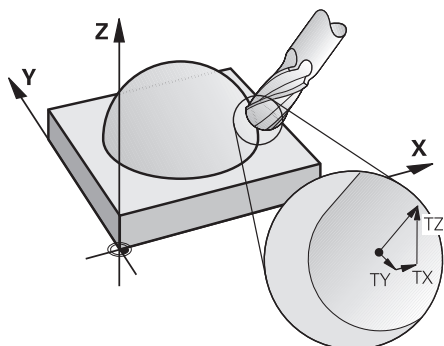
> **i**   Note the differences between face milling and peripheral milling.
> **Further information:** "3D tool compensation during face milling (#9 / #4-01-1)", Page 809
> **Further information:** "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 816

17.4.2 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 805

Requirements

■ Advanced Functions Set 2 software option (#9 / #4-01-1)

■ 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 947

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 300

In addition, the straight lines **LN** contain a surface normal vector as well as an optional tool vector.

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. If the axis position does not change, you can nevertheless program more than four axes.

Input

```
LN  X+31.737 Y+21.954 Z+33.165  NX+0.2637581 NY+0.0078922 NZ-0.8764339 TX
    +0 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 798 Optional syntax element
F, **FMAX**, **FZ**, **FU** or **F AUTO**	Feed rate **Further information:** "Feed rate F", Page 284 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.
- HEIDENHAIN recommends using normalized vectors with at least seven decimal places. This enables you to achieve high accuracy and avoid possible drops in infeed during machining operations.
- 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. The vector amount corresponds to the root of the sum of the squares of its components.

$$\sqrt{NX^2 + NY^2 + NZ^2} = 1$$

17.4.3 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 tool radius", Page 796

- Compensation in tool call

  Further information: "Tool call by TOOL CALL", Page 279

- Compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 801

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 1684

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.
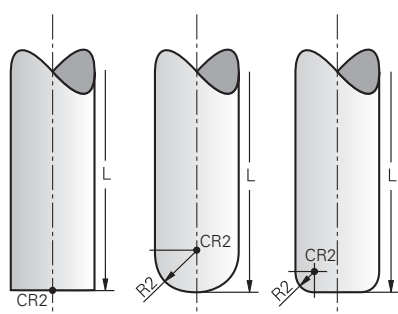
Position of CR2 with the individual tool shapes

Further information: "Presets on the tool", Page 253

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 (#9 / #4-01-1)", Page 1001

  Use the simulation to ensure that no contours are damaged by the tool oversize.

- The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.

17.4.4 3D tool compensation during face milling (#9 / #4-01-1)

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.
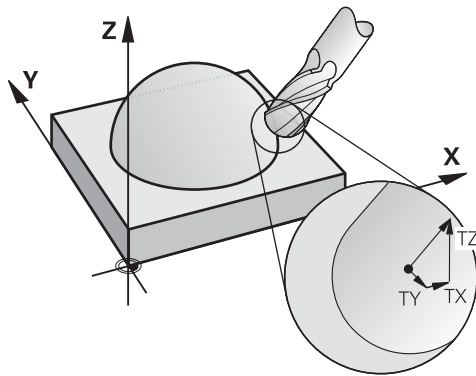
Requirements

- Advanced Functions Set 2 software option (#9 / #4-01-1)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system
  Further information: "Straight line LN", Page 806
- NC program with **M128** or **FUNCTION TCPM**
  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986
  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787

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

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.

Example

11 L X+36.0084 Y+6.177 Z-1.9209 R0	; No compensation is possible

11 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

11 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

11 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, then the control maintains the tool perpendicular to the workpiece contour.
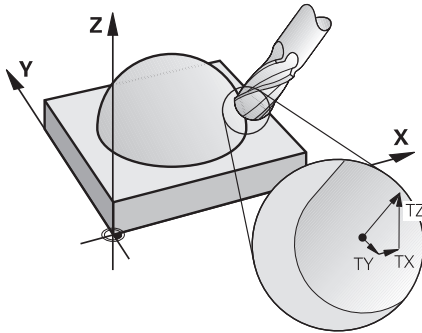
■ If a tool orientation **T** has been defined in the **LN** block and **M128** (or **FUNCTION TCPM**) is active at the same time, then the control will position the rotary axes automatically in such a way that the tool can reach the specified tool orientation. If you have not activated **M128** (or **FUNCTION TCPM**), then the TNC 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 (#9 / #4-01-1)", Page 819

Examples

Compensate re-worked ball-nose cutter
CAM output at tool tip
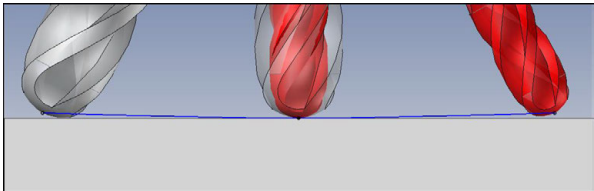
Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

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
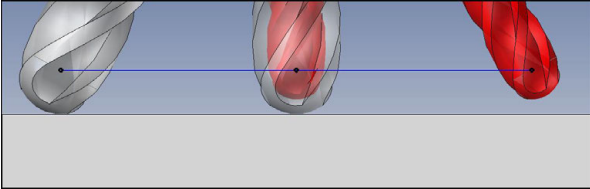
Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the ball
- 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

> **i** With TCPM **REFPNT CNT-CNT** the tool compensation values are identical for the outputs on the tool tip or center of the ball.

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 ball
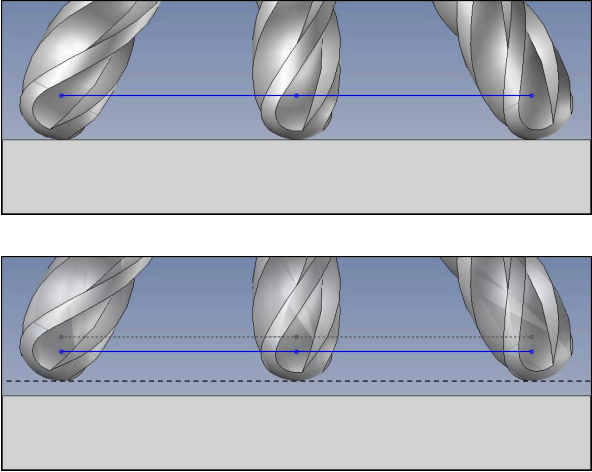
Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the ball
- 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

17.4.5   3D tool compensation during peripheral milling (#9 / #4-01-1)

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.
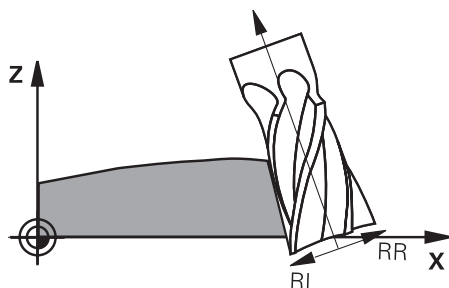
Requirements

- Advanced Functions Set 2 software option (#9 / #4-01-1)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 806
- NC program with spatial angles
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
- NC program with tool radius compensation **RL** or **RR**

  Further information: "Tool radius compensation", Page 798

Description of function

The variants below are possible with peripheral milling:

- **L** block with programmed rotary axes, **M128** or **FUNCTION TCPM** 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

The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.

Example

11 M128	
* - ...	
21 L X+48.4074 Y+102.4717 Z-7.1088 C+0 B-20.0115 RL	; Compensation is possible, compensation direction RL

11 LN X+60.6593 Y+102.4690 Z-7.1012 NX0.0000 NY0.9397 NZ0.3420 TX-0.0807 TY0 TZ0.9366 R0 M128	; Compensation is possible

11 LN X+60.6593 Y+102.4690 Z-7.1012 TX-0.0807 TY0 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 (#9 / #4-01-1)", Page 819

Example

Compensate re-worked end mill
CAM output at tool center
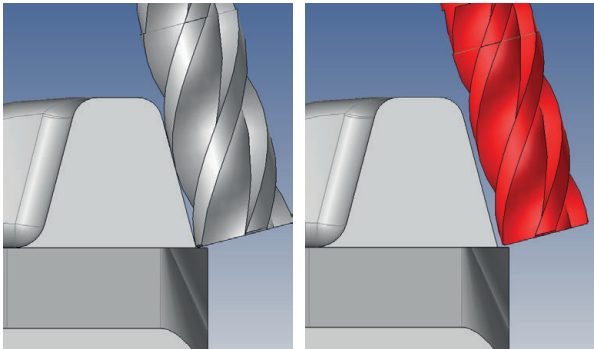
You use a re-worked Ø 11.8 mm end mill instead of Ø 12 mm.

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

17.4.6 3D tool compensation with the entire tool radius with FUNCTION PROG PATH (#9 / #4-01-1)

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 805
- Tools for 3D compensation

  Further information: "Tools for 3D tool compensation", Page 808

Requirements

- Advanced Functions Set 2 software option (#9 / #4-01-1)
- 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 947

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

18

Files

18.1 File management

18.1.1 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 function encompasses the **Files** operating mode and the workspace as well as the **Open File** windows.

Related topics

■ Data backup

Further information: "Backup and restore", Page 1829

■ Connecting network drives

Further information: "Network drives on the control", Page 1792

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** If you cut a file or a folder, then the control dims the icon of the file or the folder.
🗑	**Delete**
☆⁺	**Add favorite**
☆⁻	**Remove favorite**
⭐	Favorite If you add a favorite, then the control displays this icon next to the file or the folder.
⏏	Eject USB device
🔓	**Deactivate write-protection**
🔒	**Activate write-protection** If write protection is active, then the control displays this icon next to the file or the folder.
<eof>	With **end of file**, the control indicates that the complete file is visible in the preview area.
⬚	The control only displays a part of the file in the preview area.
New folder	Create new folder
New file	Create new file ℹ You create a new table in the **Tables** operating mode. **Further information:** "The Tables operating mode", Page 1666
File functions	The control opens the context menu. **Further information:** "Context menu", Page 1174 Only in the **Files** operating mode
Mark **CTRL + SPACE**	The control marks the file and opens the action bar. Only in the **Files** operating mode
↰ **CTRL + Z**	**Undo**

Icon, button or shortcut	Meaning
↻ **CTRL + Y**	**Redo**
Show as document	The control opens the file in the **Document** workspace. **Further information:** "The Document workspace", Page 837
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**
	  ■ Convert the format and content of files from the iTNC 530
	  ■ Modify faulty files
	  Further information: "Converting files", Page 839
	■ **Mount network share**
	  Further information: "Network drives on the control", Page 1792
	Only in the **Files** operating mode

Areas of file management
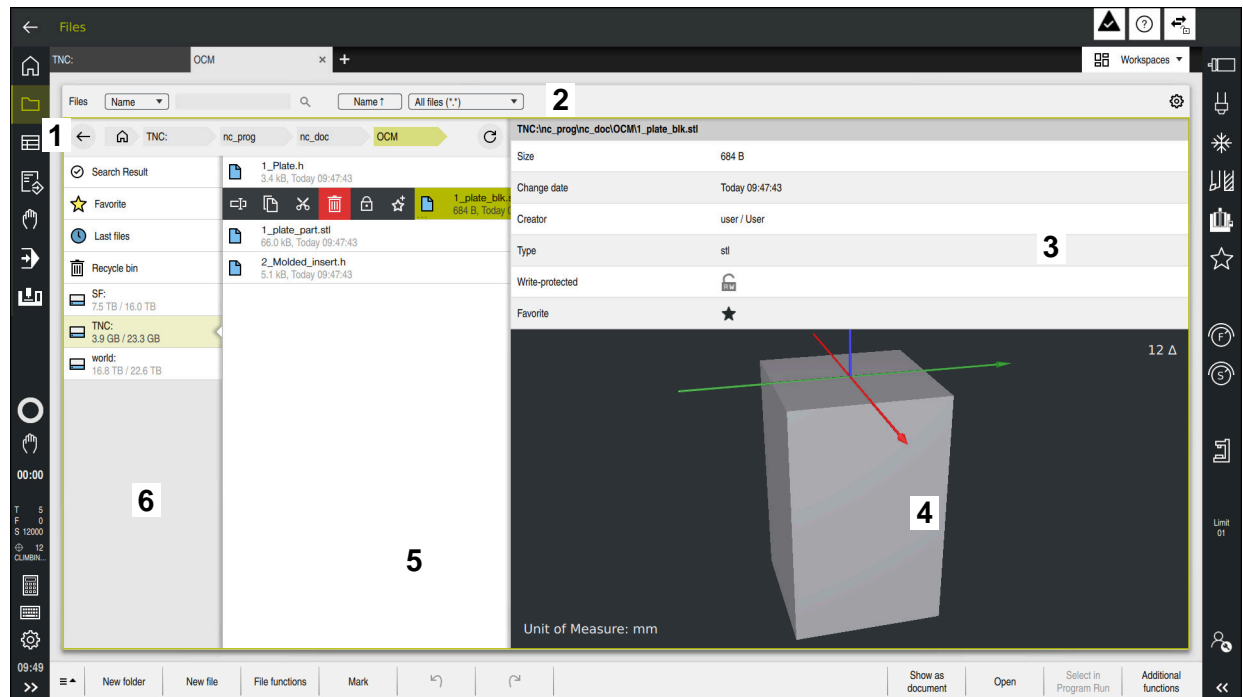
The **Files** operating mode

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 826

- Sorting

  Further information: "Sorting in the title bar", Page 826

- Filtering

  Further information: "Filtering in the title bar", Page 826

- Settings

  Further information: "Settings in the title bar", Page 826

3 Information area

Further information: "Information area", Page 827

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 for selection using the navigation column.

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

If you swipe a file or folder to the right, the control displays the following file functions:

- Rename
- Copy
- Cut
- Delete
- Activate or deactivate write protection
- Add or remove a favorite

You can also select some of these file functions with the context menu.

Further information: "Context menu", Page 1174

6   Navigation column

Further information: "Navigation column", Page 827

Full-text search in the title bar

Use the full-text search to look for any strings in the names or contents of files. Use the selection menu to choose whether the control searches the names or contents of the files.

Before a search, you first need to choose the path in which the control is to conduct the search. Based on the chosen path, the control only searches within the subordinate structure. In order to refine a search, you can search again within an existing search result.

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

The control provides standard filters for file types. If you would like to filter for other file types, then you can search using the placeholder in the full-text search function.

Further information: "Full-text search in the title bar", Page 826

Settings in the title bar

In the **Settings** window the control offers the following toggle switches:

- **Show hidden files**

  When the toggle switch is active the control shows hidden files. Names of hidden files start with a dot.

- **Show dependent files**

  When the toggle switch is active the control shows dependent files. Dependent files end with ***.dep** or ***.t.csv**.

Information area

In the information area the control shows the path of the file or folder.

Further information: "Path", Page 828

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 1174

- Drives (e.g., **TNC:**)

  The control displays internal and external drives (e.g., a USB device).

  The control displays the occupied and total memory space under each drive.

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:

Character	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 199
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
D	Table with workpiece datums **Further information:** "Datum table *.d", Page 1721
DEP	Automatically generated table with data that depend on the NC program (e.g., the tool usage file) **Further information:** "Tool usage file", Page 1702
P	Table for pallet-oriented machining **Further information:** "The Job list workspace", Page 1620
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) **Further information:** "Point table *.pnt", Page 1720
PR	Table with workpiece presets **Further information:** "Preset table *.pr", Page 1710
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 *.tab", Page 1707 **Further information:** "Cutting data calculator", Page 1181
TCH	Table with the assignment of the tool magazine **Further information:** "Pocket table tool_p.tch", Page 1698
T	Table with tools for all technologies **Further information:** "Tool table tool.t", Page 1684
TP	Table with touch probes (#17 / #1-05-1) **Further information:** "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694
TNCDRW	Contour description as a 2D drawing **Further information:** "Graphical programming", Page 1089
M3D	Format for tool carriers or collision objects (#40 / #5-03-1), for example **Further information:** "Options for fixture files", Page 857
TNCBCK	File for data backup and restoration **Further information:** "Backup and restore", Page 1829
EXP	Configuration file for saving and importing configurations of the control interface **Further information:** "Configuring the control's user interface", Page 1838

The control opens these file types with an internal application or with a HEROS tool.

Further information: "Opening files with additional software", Page 1881

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 270
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 1209
DXF	2D CAD files
IGS/IGES STP/STEP	3D CAD files **Further information:** "Opening CAD files with CAD Viewer", Page 1107
CHM	Help files in compiled or compressed format
CFG	Configuration files of the control **Further information:** "Options for fixture files", Page 857 **Further information:** "Machine parameters", Page 1833
CFT	3D data of a parameterizable tool-carrier template **Further information:** "Tool carrier management", Page 273
CFX	3D data of a geometrically determined tool carrier **Further information:** "Tool carrier management", Page 273
HTM/HTML	Text file with structured content of a website that can be opened in a browser (e.g., the integrated product aid) **Further information:** "User's Manual as integrated product aid: TNCguide", Page 66
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 1881
INI	Initialization file (e.g., can contain program settings)
A	Format file (e.g., for defining the screen output format in connection with **FN 16**)
TXT	Text file (e.g., for saving the results of measurement cycles in connection with **FN 16**)
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 1868
OGG	Container file format for the OGA, OGV, and OGX media types

File type	Application
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 1881

Notes

- The control has 21 GB of disk space. The maximum size of any file is limited to 2 GB.

- When you open an NC program, the control requires free disk space that is three times the file size of the NC program.

- When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

  In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.

  Further information: "The Tables operating mode", Page 1666

- 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 1066

- 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 287

- 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.

- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

Hints about copied files

- If you copy a file and then paste it to the same folder, the control adds the suffix **_1** to the file name. The control increments the number sequentially for each consecutive copy.
- 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.

18.1.2   The Open File workspace

Application

In the **Open File** workspace you select or create files, for example.

Description of function

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	The Tables operating mode	The Editor operating mode	The Program Run operating mode
New folder	✓	✓	–
New file	✓	✓	–
Open	✓	✓	✓

18.1.3 Quick selection workspaces

Application

In the **Quick selection new table** and **Quick selection new file** workspaces, you can create files or open existing files, depending on the active operating mode.

Description of function

You can open the workspaces by using the **Add** function in the operating modes below:

■ **Tables**

Further information: "Quick selection new table workspace", Page 835

■ **Editor**

Further information: "Quick selection new file workspace", Page 836

Further information: "Icons on the control's user interface", Page 107

Quick selection new table workspace

The **Quick selection new table** workspace makes the following buttons available:

■ **Create new table**

Further information: "The Create new table window", Page 1668

■ **Tool management**

■ **Pocket table**

■ **Presets**

■ **Touch probes**

■ **Datums**

■ **T usage order**

■ **Tooling list**

The **Quick selection new table** workspace contains the following areas:

■ **Active tables for machining**

■ **Active tables for simulation**

The control displays the **Presets** and **Datums** buttons in both areas.

With the **Presets** and **Datums** buttons, you can open the table that is active in the program run or in the simulation. If the same table is active in program run and the simulation, then the control opens this table only once.

Quick selection new file workspace

The **Quick selection new file** workspace offers the following buttons:

Area	Button
New NC program	■ **NC program mm** ■ **NC program inch** ■ **ISO program mm** ■ **ISO program inch** **Further information:** "Programming fundamentals", Page 199
New graphical programming	**Contour** **Further information:** "Graphical programming", Page 1089
New text file	■ **Text file** with a ***.txt** extension ■ **Format file** with an ***.a** extension **Further information:** "The Text editor workspace", Page 839
New job	**Job list** **Further information:** "The Job list workspace", Page 1620

18.1.4 The Document workspace

Application

You can open files for viewing in the **Document** workspace, for example a technical drawing.

Related topics

- Supported file types
  Further information: "File types", Page 829
- **Show as document** button in the **Files** operating mode
  Further information: "Icons and buttons", Page 823

Description of function

The **Document** workspace is available in every operating mode and application. If you open a file, then the control displays the same file in all operating modes.

Further information: "Overview of the operating modes", Page 93

The control shows the file path in the file information bar.

You can open the following file types in the **Document** workspace:

- PDF files
  The **Document** workspace makes a search function available for PDF files.
- HTML files
- Text files, such as *.txt
- Image files, such as *.png
- Video files, such as *.webm

Further information: "File types", Page 829

You can, for example, transfer dimensions from a technical drawing using the clipboard in the NC program.

Icons in the Document workspace

The following icons are shown in the **Document** workspace:

Icon	Meaning
🗁	**Open File** **Further information:** "Open file", Page 838
⊕	Open or close the **Internet** window The **Internet** window allows entering and calling a URL. You may also bookmark the URL.
← →	**Navigate** Navigate between the last opened files
↻	**Refresh** (e.g., log file or a touch probe cycle)

When a PDF file is open, the **Document** workspace additionally displays the following icons:

Icon	Meaning
✋	Activate or deactivate **Move** If this icon is active, highlighting text with the mouse is not possible. Instead, the visible area can be shifted in any direction with the mouse.
⌃ ⌄	**Navigate** Select the previous or the next element Depending on the position of icons, you either navigate between the file pages or the search results.
Page X/X	Current page number and total number of pages
100%	Current size of content Open or close the **Scale** select menu
🔍	**Reset scaling** Scaling the content to the full width
↺ ↻	**Rotate** Rotate the content by 90° anti-clockwise or clockwise

Open file

To open the file in the **Document** workspace:

▶ If applicable, open the **Document** workspace

🗁 ▶ Select **Open File**
> The control opens a selection window with the file manager.
▶ Select the desired file
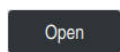
[Open] ▶ Select **Open**
> The control displays the file in the **Document** workspace.

18.1.5 The Text editor workspace

Application

Use the **Text editor** workspace to create and edit text files.

Related topics
- File types
  Further information: "File types", Page 829
- Displaying text files in the **Document** workspace
  Further information: "The Document workspace", Page 837

Description of function

The **Text editor** workspace is available in the **Editor** operating mode.

The following file types can be edited in the **Text editor** workspace:
- Text files, such as ***.txt**
  Example: measuring logs output with **FN 16**
- Text files, such as ***.a**
  Example: format file for **FN 16**

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1028

Further information: "File types", Page 829

> **ℹ** Refer to your machine manual.
> The machine manufacturer can define further file types that you can edit in the text editor.

Icons in the Text editor workspace

The following icons are shown in the **Text editor** workspace:

Icon	Meaning
1	Unhide or hide the **Line number**
⟹	Activate or deactivate the **Line number**
	When activating the **Line number**, the control will automatically add line breaks in the text.

18.1.6 Converting files

Application

In order to use a file created on the iTNC 530 on the TNC7 basic as well, the control must adapt the file's format and content. Use the **Update TAB / PGM** function for this.

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 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 _

If you convert tables from an earlier control using the **Update TAB / PGM** function, then the control makes the following changes as needed:

- The control changes decimal commas into decimal points.
- The control adopts all supported tool types and assigns the **Undefined** type to all unknown tool types.

The **Update TAB / PGM** function also allows you to adapt tables of the TNC7 basic if necessary.

Further information: "Tool table tool.t", Page 1684

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

▶ 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

> ### *NOTICE*
>
> **Caution: Data may be lost!**
>
> If you use the **Update TAB / PGM** function, then data may be irrevocably deleted or altered!
>
> ▶ Create a backup copy prior to converting the file

- The machine manufacturer uses import and update rules to define which adaptations the control is to execute, such as umlaut removal.
- 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.

18.1.7 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

  > ℹ️ The control does not support USB devices with other file systems, such as NTFS.

- A ready data interface
  Further information: "Serial data transfer", Page 1873

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 1791
- 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 1881

18.2 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 permits, for example, 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 829

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

> **i** 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 822

Input

| 11 OPEN FILE "FILE1.PDF" STOP |

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Selection** ▶ **OPEN FILE**

The NC function includes the following syntax elements:

Syntax element	Meaning
OPEN FILE	Syntax initiator for the OPEN FILE function
File or **QS**	Path of the file to be opened Fixed or variable path Selection by means of a selection window
STOP	Interrupts the program run or simulation Optional syntax element

Copying, moving 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.

Input

Copying a file

11 FUNCTION FILE COPY "FILE1.PDF" TO "FILE2.PDF"	; Copy the file from the NC program

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ FUNCTION FILE ▶ FUNCTION FILE COPY

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE COPY	Syntax initiator for the Open file function
File or **QS**	Path of the file to be copied Fixed or variable path Selection by means of a selection window
TO File or **QS**	Path of the file to be substituted Fixed or variable path Selection by means of a selection window

Moving a file

11 FUNCTION FILE MOVE "FILE1.PDF" TO "FILE2.PDF"	; Move the file from the NC program

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ FUNCTION FILE ▶ FUNCTION FILE MOVE

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE MOVE	Syntax initiator for the Move file function
File or **QS**	Path of the file to be relocated Fixed or variable path Selection by means of a selection window
TO File or **QS**	Path of the file to be substituted Fixed or variable path Selection by means of a selection window

Deleting a file

11 FUNCTION FILE DELETE "FILE1.PDF"	; Delete the file from the NC program

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ FUNCTION FILE ▶ FUNCTION FILE DELETE

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE DELETE	Syntax initiator for the Delete file function
File or **QS**	Path of the file to be deleted Fixed or variable path Selection by means of a selection window

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.

19

Collision Monitoring

19.1 Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)

Fundamentals

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.
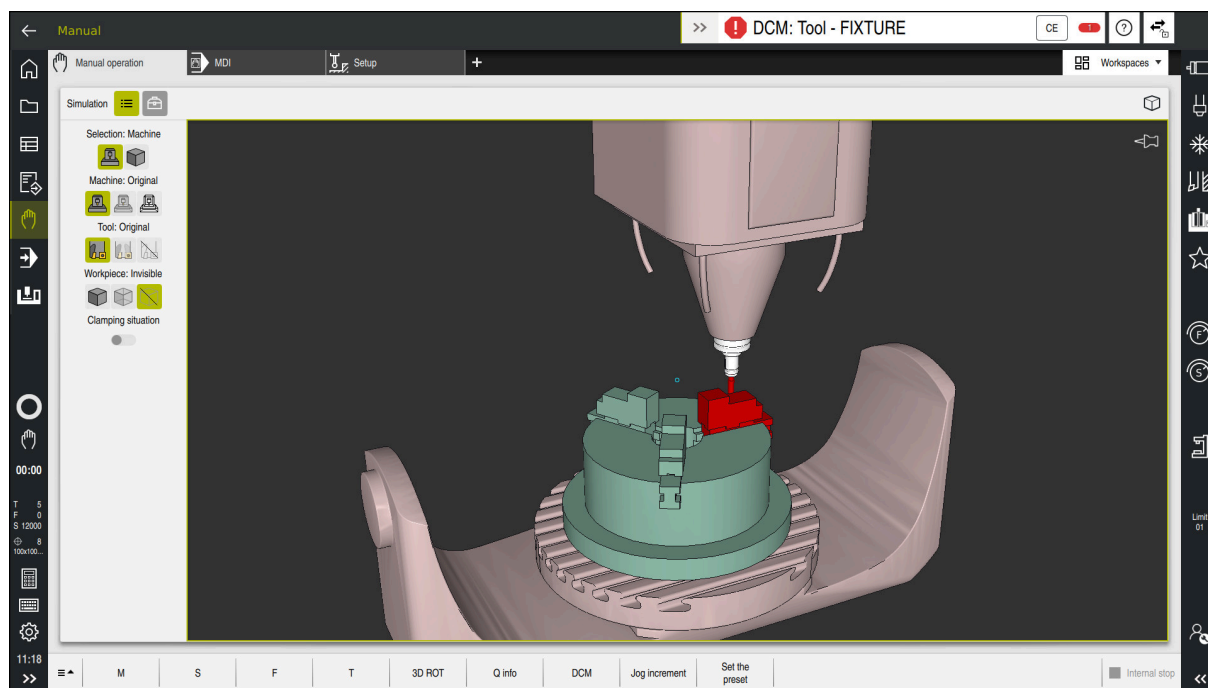
Dynamic Collision Monitoring (DCM) including collision warning

Related topics

- Fundamentals of fixture management

  Further information: "Fixture management", Page 856
- Extended tests in the simulation

  Further information: "Advanced checks in the simulation", Page 880
- Fundamentals of tool carrier management

  Further information: "Tool carrier management", Page 273
- Reduce the minimum clearance between two collision objects (#140 / #5-03-2)

  Further information: "Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)", Page 878

Requirements

- Dynamic Collision Monitoring (DCM) software option (#40 / #5-03-1)
- 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 management", Page 856
- Tools with a positive radius **R** and length **L**.

  Further information: "Tool table tool.t", Page 1684
- The values in the tool management equal the actual tool dimensions

  Further information: "Tool management ", Page 269

Description of function

> ⚙ Refer to your machine manual.
> The machine manufacturer adapts the Dynamic Collision Monitoring (DCM) function to the control.

The machine manufacturer can define machine components and minimum distances 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.

| >> | 🛑 DCM: Tool - FIXTURE | CE |

Error message for Dynamic Collision Monitoring (DCM)

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk 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 **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 will not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

- ▶ Activate the **Advanced checks** toggle switch for the simulation
- ▶ 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 880

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 853

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 clearance. 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 853

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.

> **i**   The enhanced collision monitoring shows collisions between the workpiece and tools or tool holders.
>
> **Further information:** "Advanced checks in the simulation", Page 880

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: "The Visualization options column", Page 1200

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.
- Handwheel override (#21 / #4-02-1) is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes

NOTICE
Danger of collision! If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk 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 **Single Block** mode while DCM is deactivated

To Dynamic Collision Monitoring (DCM) for the **Manual** and **Program Run** operating modes:

🖐	▶ Select the **Manual** operating mode
	▶ Select the **Manual** application
DCM	▶ Select **DCM**
	> The control opens the **Dyna. Coll. Monitoring (DCM)** window.
	▶ Activate DCM in the desired operating modes, using the toggle switches
OK	▶ Press **OK**
	> The control activates DCM in the selected operating modes.

ℹ 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.

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** toggle switch
	> The control activates DCM in the **Editor** operating mode.

ℹ The control displays the status of Dynamic Collision Monitoring (DCM) in the **Simulation** workspace.
Further information: "Icons in the Simulation workspace", Page 1199

Activating the graphic display of the collision objects
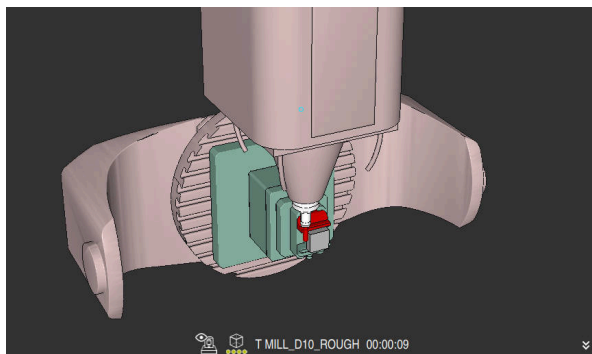
Simulation in the **Machine** mode

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**)

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.

19.1.1 Deactivating or activating the DCM NC function in the NC program with FUNCTION DCM

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.

Related topics

■ Reduce the minimum clearance between two collision objects (#140 / #5-03-2)

Further information: "Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)", Page 878

Requirement

■ Dynamic Collision Monitoring (DCM) is active for the **Program Run** operating mode

Description of function

NOTICE
Danger of collision! If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk 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 **Single Block** mode while DCM is deactivated

FUNCTION DCM is only in effect 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 clearance 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 FUNCTION DCM

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

19.2 Fixture management

19.2.1 Fundamentals

Application

You can integrate fixtures as 3D models in the control in order to represent clamping situations for simulation or execution.

When DCM is active, the control checks during simulation or machining if the fixture collides (#40 / #5-03-1).

Related topics

■ Dynamic Collision Monitoring (DCM (#40 / #5-03-1))

Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

■ Integrating an STL file as workpiece blank

Further information: "STL file as workpiece blank with BLK FORM FILE", Page 249

Requirements

■ 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 it into the control
  Further information: "Options for fixture files", Page 857
- Fixture placement
  - The **Set up fixtures** function in the **Setup** (#140 / #5-03-2) application
    Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859
  - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 869
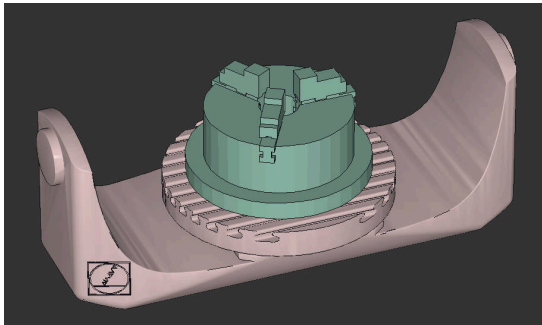
Three-jaw chuck loaded as fixture

Options for fixture files

If you use the **Set up fixtures** function to integrate fixtures, then only STL files are possible (#140 / #5-03-2).

Alternatively, CFG and M3D files can be set up manually.

You can use the function **3D mesh** (#152 / #1-04-1) 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 (#152 / #1-04-1)", Page 1124

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 datum clamping systems and recurring setups.

If an STL file does not meet the requirements of the control, then the control issues an error message.

With the software option CAD Model Optimizer (#152 / #1-04-1), you can adapt STL files that do not meet the requirements and then use them for fixtures.

Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124

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 in effect on the control. **KinematicsDesign** can be used to create and edit CFG files on the control.

Further information: "Editing CFG files with KinematicsDesign", Page 870

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.

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**).
- When DCM is active, the control checks during simulation or machining if the fixture collides (#40 / #5-03-1).
  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:
  HEIDENHAIN NC solutions
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

19.2.2 Integrating fixtures into collision monitoring (#140 / #5-03-2)

Application

The **Set up fixtures** function determines the position of a 3D model in the **Simulation** workspace, matching the real fixture in the workspace. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- The **Simulation** workspace

  Further information: "The Simulation Workspace", Page 1197
- Dynamic Collision Monitoring (DCM)

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
- Fixture monitoring

  Further information: "Fixture management", Page 856
- Setting up a workpiece with graphical support (#159 / #1-07-1)

  Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 1277

Requirements

- If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.
  Further information: "Calibrating the workpiece touch probe", Page 1272

  If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.
- Software option Dynamic Collision Monitoring (DCM) version 2 (#140 / #5-03-2)
- Workpiece touch probe
- Permitted fixture file matching the real fixture

  Further information: "Options for fixture files", Page 857

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 direction, the control changes the status of that axis from red to green.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.

Further information: "Error estimate diagram", Page 864

The scope of the **Set up fixtures** function depends on the Extended Functions Group 1 (#8 / #1-01-1) and Extended Functions Group 2 (#9 / #4-01-1) software options as follows:

- Both software options enabled:

  You can tilt before probing, and incline the tool while probing, in order to probe even complex fixtures.

- Only Extended Functions Group 1 (#8 / #1-01-1) enabled:

  You can tilt before probing. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.

  > **ⓘ** If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.

- None of the two software options is enabled:

  You cannot tilt before probing. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736

Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787

Extension of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphic support for setting up the fixture.
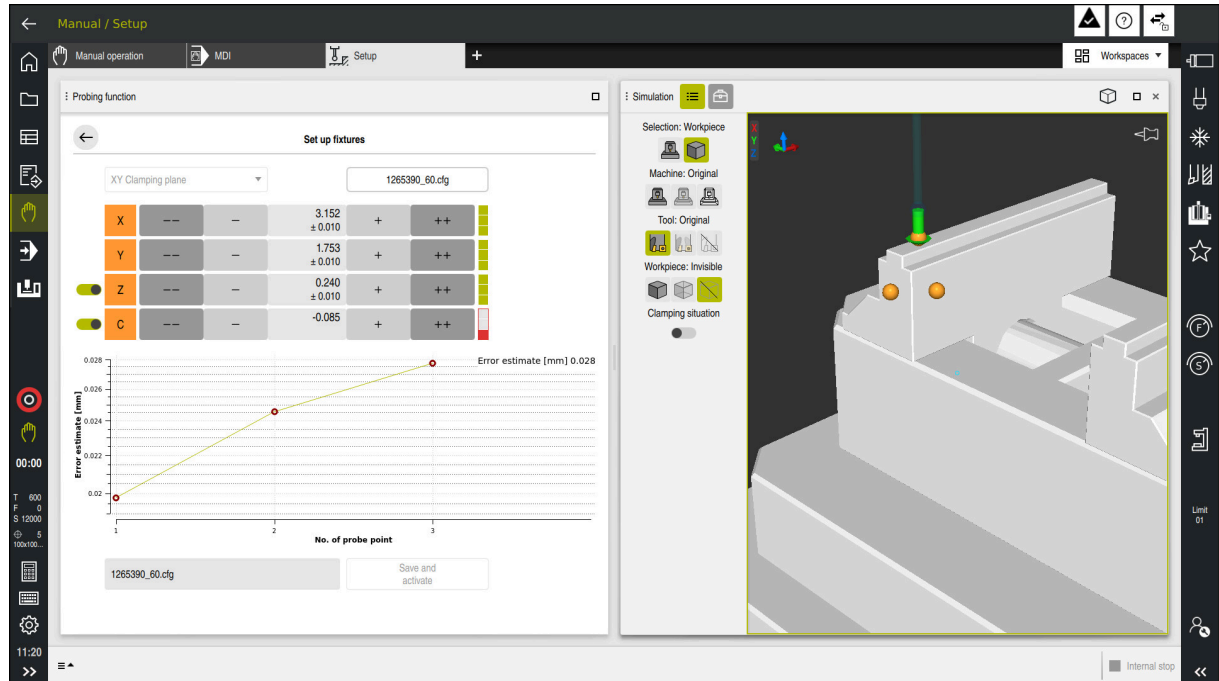
The **Set up fixtures** function with the **Simulation** workspace open

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.
    In this case, you can adjust the position of the 3D model in the simulation, if applicable.
  - Red arrow
    Probing in the direction of the arrow is not possible.

    > **i** 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 under certain conditions. Probing is done in a deselected direction or might cause collisions.
  - Green arrow
    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Meaning
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 **ℹ** Depending on the selected clamping plane, the control displays the corresponding axis directions. In the **XY Clamping plane**, for example, the control displays the axes **X**, **Y**, **Z** and **C**.
1265390_60.cfg	Name of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited before saving.
– –	Shifts the position of the virtual fixture by 10 mm or 10° in the negative axis direction **ℹ** 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
-15.982 ± 0.017	■ Enter the position of the virtual fixture directly ■ Value and estimated accuracy after probing
+	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 direction is deselected for this set-up process and will not be taken into account. ■ White No probing points have been determined yet. ■ Red The control cannot determine the fixture position in this axis direction. ■ Yellow The position of the fixture in this axis direction already contains information. The information is not meaningful yet. ■ Green The control can determine the fixture position in this axis direction.
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM).

Icon or button	Meaning
ℹ	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 next to the button.

When using a datum clamping system and for this reason you do not want to consider one axis direction (such as **Z**) when setting up the fixture, the axis in question can be deselected by a toggle switch. The control will not take deselected axis directions into account in the set-up process and positions the fixture by considering the remaining axis directions only.

Error estimate diagram

Every probing point further restricts the possible positioning of the fixture and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture. The control not only considers the probing points, but also the entire fixture.

As soon as the error estimate diagram shows green circles and the desired accuracy, the set-up process is completed.

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)
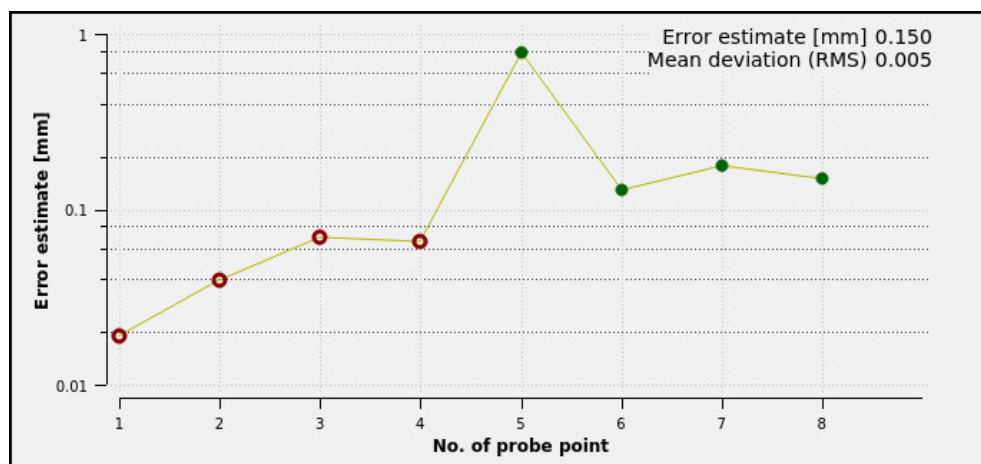
Error estimate diagram in the **Set up fixtures** function

The error estimate diagram of the **Set up fixtures** function displays the following information:

- **Mean deviation (RMS)**

  This area shows the average distance of the measured probing points from the 3D model in mm.

- **Error estimate [mm]**
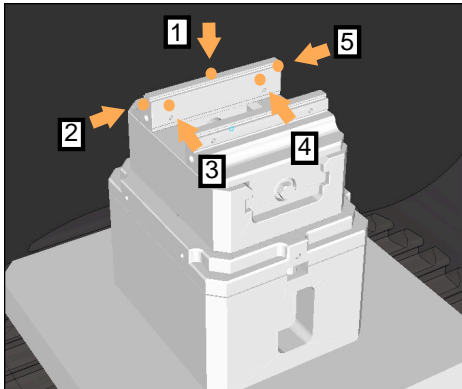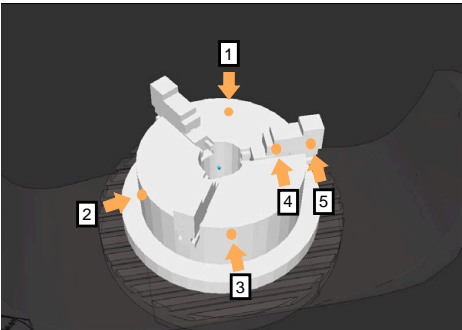
  This axis shows the course of the revised model position by means of the individual probing points. Red circles are shown until the values for all axis directions are determined. From then on, the control displays green circles.

- **No. of probe point**

  This axis shows the numbers of the individual probing points.

Example of sequence of fixture probing points

Below are some of the probing points that can 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

ℹ The desired 3D model must meet the requirements of the control.
Further information: "Options for fixture files", Page 857

To measure a vise 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 notable point

            ℹ 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

            ℹ 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, for example, 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

            ℹ 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.

| Save and activate |

▶ 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 probe the clamping situation in the machine exactly, 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 870
- 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 870
- 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: "Load and remove fixtures with the FIXTURE NC function", Page 869

19.2.3 Load and remove fixtures with the FIXTURE NC function

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 management", Page 856

Requirement

- A measured fixture file exists

Description of function

When DCM is active, the control checks during simulation or machining if the fixture collides (#40 / #5-03-1).

The **FIXTURE SELECT** function selects a fixture by means of a pop-up window.

The **FIXTURE RESET** function removes the fixture.

Input

| 11 FIXTURE SELECT "TNC:\system \Fixture\JAW_CHUCK.STL" | ; Load the fixture as an STL file |

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ FIXTURE

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
	Fixed or variable path
	Selection by means of a selection window
	Only if **SELECT** has been selected

Note

For optimum performance, HEIDENHAIN recommends CFG files that contain no more than 20,000 triangles.

19.2.4 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.

Related topics

- Combine fixtures into complex clamping arrangements
  Further information: "Combining fixtures in the New Fixture window", Page 875

Description of function

When opening a CFG file in the control, the control makes **KinematicsDesign** available as a selection item.

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.
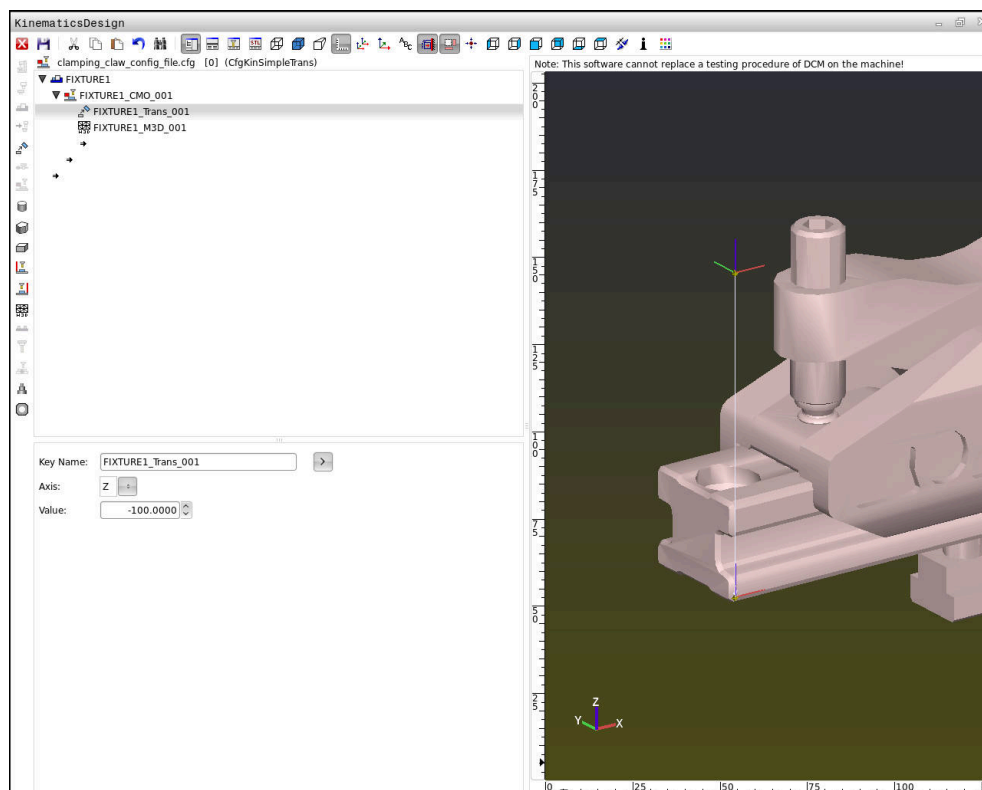
Syntax in CFG files

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 ℹ 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**.
`CfgKinFixModel(key:="Fix_Model",`	Fixture designation

Function	Description
`kinObjects:=["fixture"])`	**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 chucking equipment**

> **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.

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**
M3D   > 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.

Notes

- If one of the transformations contains the **?** character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

  Further information: "Combining fixtures in the New Fixture window", Page 875

- 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 first 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 subsequent components 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

19.2.5 Combining fixtures in the New Fixture window

Application

The **New Fixture** window allows combining several fixtures and saving them as a new fixture. This enables realizing and monitoring complex clamping situations.

Related topics

- Fundamentals of fixtures

  Further information: "Fundamentals", Page 856

- Integrating fixtures into the NC program

  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 869

- Set up fixtures (#140 / #5-03-2)

  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859

Requirement

- Fixtures of suitable format:
  - STL file
    - 20,000 triangles maximum
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

Description of function

To navigate to this function:

Tools ▶ Combine fixtures

The control also makes this function available as a selection option for opening CFG files.
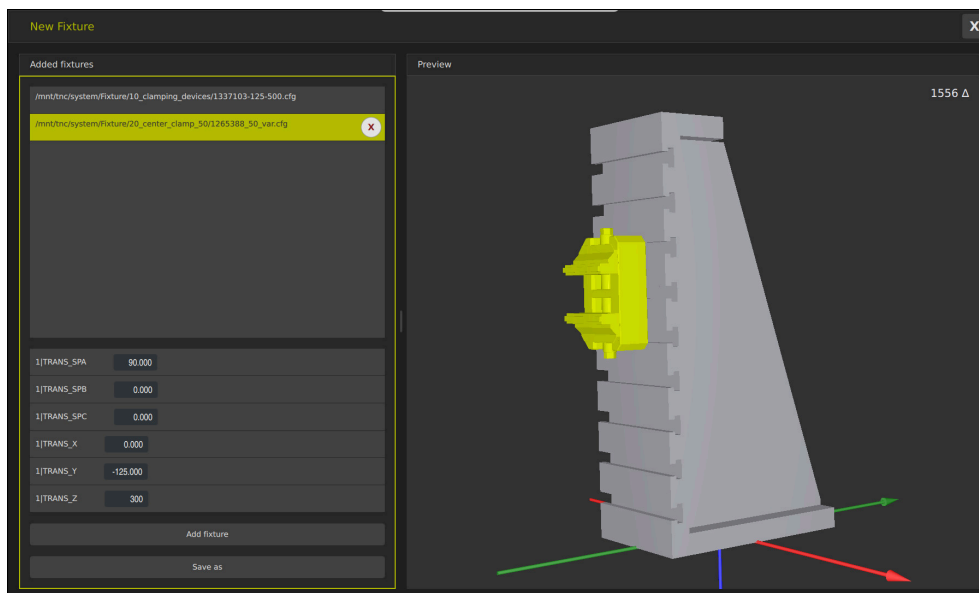
Combined fixture with variable transformations

The **Add fixture** button selects all required fixtures one by one.

If one of the transformations contains the **?** character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

The control displays a preview of the combined fixture and the total number of all triangles.

The **Save as** button saves the combined fixture as a CFG file.

Notes

- For optimum performance, HEIDENHAIN recommends that combined fixtures contain no more than 20,000 triangles.
- If the position or the size of a fixture must be adapted, use **KinematicsDesign**.
  Further information: "Editing CFG files with KinematicsDesign", Page 870

19.2.6   Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)

Application

Some machining steps are by design performed close to a fixture. If Dynamic Collision Monitoring (DCM) is active and the distance between the fixture and tool falls below the defined minimum clearance, the control issues an error message and stops the movement.

To enable using DCM in such machining steps, the control makes the **FUNCTION DCM DIST** NC function available. This NC function allows reducing the permitted minimum clearance between the tool and the fixture within a NC program.

Related topics

■ Dynamic Collision Monitoring (DCM (#40 / #5-03-1))

Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

■ Loading and removing the fixture

Further information: "Load and remove fixtures with the FIXTURE NC function", Page 869

Requirements

■ Software option Dynamic Collision Monitoring (DCM) version 2 (#140 / #5-03-2)
■ Dynamic Collision Monitoring (DCM) is active

Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

■ Fixture is integrated in the NC program

Further information: "Load and remove fixtures with the FIXTURE NC function", Page 869

Description of function

When **FUNCTION DCM DIST** is active, the control displays an icon in the **Positions** workspace and in the information bar. The **Simulation** workspace displays the collision objects in question in orange.

The control resets **FUNCTION DCM DIST** with the following NC functions:

■ **FUNCTION DCM DIST RESET**
■ **M2** or **M30**

Input

11 FUNCTION DCM DIST FIXTURE1	; Reduce the minimum clearance to 1 mm

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **FUNCTION DCM DIST**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DCM DIST	Syntax opener for reducing the minimum clearance between the fixture and the tool
FIXTURE or **RESET**	Reduce the minimum clearance or reactivate the minimum clearance defined by the machine manufacturer Fixed or variable number Input: **0.0000...2.0000**

Notes

> **NOTICE**
>
> **Danger of collision!**
>
> If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk 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 **Single Block** mode while DCM is deactivated

---

> **NOTICE**
>
> **Danger of collision!**
>
> The **FUNCTION DCM DIST** NC function may lead to collisions, such as during CAM-generated short movements near the fixture. Dynamic Collision Monitoring (DCM) does not detect these collisions.
>
> ▶ Use **FUNCTION DCM DIST** only when needed
> ▶ Set the minimum clearance as small as necessary and as large as possible
> ▶ Check the simulation with the **Fixture collision** toggle switch active
> ▶ As an alternative, verify NC program points in question in the **Single Block** mode

---

The control cannot approach the reduced minimum clearance with the **RESTORE POSITION** function. If the approach position is closer than the minimum clearance defined by the machine manufacturer, the control will display an error message.

19.3 Advanced checks in the simulation

Application

The **Advanced checks** function allows checking in the **Simulation** workspace if collisions will occur between, for example, the workpiece and the tool.

Related topics

■ Collision monitoring of machine components by means of the Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) function

Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

Description of function

The **Advanced checks** function can be used only in the **Editor** operating mode.

When activating the **Advanced checks** toggle switch, the control opens the **Advanced checks** window.

The **Advanced checks** window allows activating the following tests:

■ **Rapid-traverse cut**

The control displays a warning in case material is removed at rapid traverse. The control displays material removal at rapid traverse in red in the simulation.

■ **Workpiece collision**

The control displays a warning in case of collisions between the tool carrier or tool shank and the workpiece.

■ **Fixture collision**

The control displays a warning in case of collisions between the tool and the workpiece fixture.

The control also considers inactive steps of a stepped tool.

You can activate several test at the same time.

Further information: "The Visualization options column", Page 1200
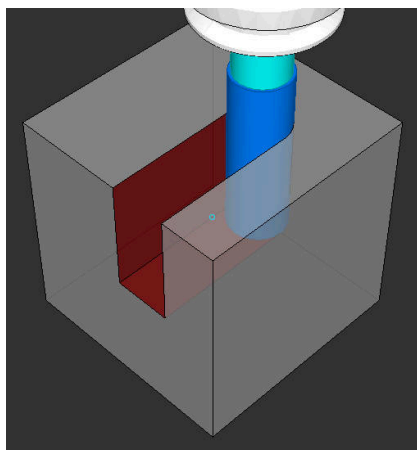
Material removal at rapid traverse

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. Even if several workpieces are clamped in the machine, the control can monitor only the active workpiece blank!

  Further information: "Defining a workpiece blank with BLK FORM", Page 244

19.4 Automatic tool liftoff with FUNCTION LIFTOFF

Application

The tool retracts from the contour by up to 2 mm. The control calculates the liftoff 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 996
- Liftoff in the tool axis with **M140**

  Further information: "Retracting in the tool axis with M140", Page 993

Requirements

- Function enabled by the machine manufacturer

  In machine parameter **on** (no. 201401), the machine manufacturer defines whether automatic liftoff 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 have the following options for programming the LIFTOFF function:

- **FUNCTION LIFTOFF TCS X Y Z**: Liftoff in the tool coordinate system (**T-CS**) with the vector resulting from **X**, **Y** and **Z**
- **FUNCTION LIFTOFF ANGLE TCS SPB**: Liftoff in the tool coordinate system (**T-CS**) with a defined spatial angle
- **FUNCTION LIFTOFF RESET**: NC function reset

Further information: "Tool coordinate system T-CS", Page 703

The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

Input

11 FUNCTION LIFTOFF TCS X+0 Y+0.5 Z +0.5	; Liftoff with the defined vector upon NC stop or power failure
12 FUNCTION LIFTOFF ANGLE TCS SPB +20	; Liftoff with spatial angle **SPB** +20 upon NC stop or power failure

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **FUNCTION LIFTOFF**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION LIFTOFF	Syntax initiator for an automatic liftoff
TCS, **ANGLE** or **RESET**	Define the liftoff direction as a vector or a spatial angle 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 liftoff in the direction of the active tool axis.

Notes

- The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the liftoff direction. If you program **M148**, the control will automatically liftoff the tool in the direction defined by the **FUNCTION LIFTOFF** function.

- In case of an emergency stop, the control will not liftoff the tool.

- The liftoff movement will not be monitored by Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum liftoff height.

- In machine parameter **feed** (no. 201405), the machine manufacturer defines the speed of liftoff movement.

20

Control Functions

20.1 Adaptive feed control (AFC) (#45 / #2-31-1)

20.1.1 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 (#45 / #2-31-1)", Page 1734

Requirements

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))
- Enabled 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 1734
- Define settings for AFC for each tool in the tool management

  Further information: "Tool table tool.t", Page 1684
- Define AFC in the NC program

  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 887
- Define AFC in the **Program Run** operating mode with the **AFC**toggle switch.

  Further information: "The AFC toggle switch in the Program Run operating mode", Page 889
- Prior to automatic control, determine the reference spindle power with a teach-in cut

  Further information: "AFC teach-in cut", Page 890

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: "The Positions workspace", Page 149

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: "The AFC tab (#45 / #2-31-1)", Page 159

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 892

■ Protection of the machine's mechanical elements

Timely feed rate reduction and shutdown reactions 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 define 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 1734

■ ***.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**. The string **<name>** is identical to 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 1737

■ ***.H.AFC2.DEP**

During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you are perfoming 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 1739

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of the active NC program.

Further information: "Editing the tables for AFC", Page 1740

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 is about to fall below the **FMIN** value, stop the machining operation (instead of deactivating the AFC function)
>
> ▶ Define the overload response 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 response.
  ■ If, with the reference spindle load, the value falls below the minimum feed factor

    The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.

    Further information: "Basic AFC settings in AFC.tab", Page 1734
  ■ 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**.
■ In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: "Areas of file management", Page 825

20.1.2 Activating and deactivating AFC

NC functions for AFC (#45 / #2-31-1)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))
- Control settings defined in the **AFC.tab** table
  Further information: "Basic AFC settings in AFC.tab", Page 1734
- Desired control setting defined for all tools
  Further information: "Tool table tool.t", Page 1684
- **AFC** toggle switch active
  Further information: "The AFC toggle switch in the Program Run operating mode", Page 889

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 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

- 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!

The AFC toggle switch in the Program Run operating mode

Application

The **AFC** toggle 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 (#45 / #2-31-1)", Page 887

Requirements

■ Adaptive Feed Control software option (AFC (#45 / #2-31-1))
■ Enabled 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** toggle switch must be activated for the NC functions for AFC to have an effect.

If you do not specifically deactivate AFC using the toggle switch, AFC remains active. The control remembers the setting of the toggle switch even if the control is restarted.

If the **AFC** toggle 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: "The Positions workspace", Page 149

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 the feed rate is about to fall below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
>
> ▶ Define the overload response 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%. Then you can no longer change the spindle speed.
- If Adaptive Feed Control is active in **Control** mode, the control assumes 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** toggle 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.

20.1.3 AFC teach-in cut

Fundamentals

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 1684
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function

  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 887

Requirements

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))
- Control settings defined in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 1734
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 1684
- Desired NC program selected in the **Program Run** operating mode
- **AFC** toggle switch active

  Further information: "The AFC toggle switch in the Program Run operating mode", Page 889

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 1737

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.

The AFC settings button

Application

The **AFC settings** button in the **Program Run** operating mode allows terminating a teach-in cut or opening the tables for AFC.

Related topics

- Fundamentals for the teach-in cut

  Further information: "Fundamentals", Page 890
- Tables for AFC

  Further information: "Tables for AFC (#45 / #2-31-1)", Page 1734

Requirements

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))
- Enabled 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

This button offers the following select options:

Button	Meaning
AFC.TAB	Editing the factory default settings When selecting this button, the control will open the **AFC.TAB** table in the **Tables** operating mode. **Further information:** "Basic AFC settings in AFC.tab", Page 1734
AFC.DEP	Editing the settings file for teach-in cuts When selecting this button, the control will open the **AFC.DEP** table for the current NC program in the **Tables** operating mode. **Further information:** "AFC.DEP settings file for teach-in cuts", Page 1737
AFC2.DEP	Editing the log file for evaluation When selecting this button, the control will open the **AFC2.DEP** table for the current NC program in the **Tables** operating mode. **Further information:** "Log file AFC2.DEP", Page 1739
Stop Teach	Terminating a teach-in cut ■ The control terminates the teach-in cut and changes to control mode **Further information:** "AFC teach-in cut", Page 890 ■ In the **AFC.DEP** table, the control changes the status of the **ST** column from teaching-in (**L**) to controlling (**C**). **Further information:** "AFC.DEP settings file for teach-in cuts", Page 1737 ■ In the **Positions** workspace, the control changes the icon for the teaching-in cut into the control mode icon. **Further information:** "The Positions workspace", Page 149

> **i** In a milling operation, you do not have to run the entire machining step in teaching-in mode. If the cutting conditions do not change significantly, you can switch to control mode immediately.

20.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear or breakage. To do this, use columns **AFC-OVLD1** or **AFC-OVLD2** in the tool management.

Related topics

■ **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management

Further information: "Tool table tool.t", Page 1684

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 1734

Tool wear and tool breakage cannot be monitored at the same time. If the **AFC_OVLD2** column contains a value, the control will ignore the **AFC_OVLD1** column.

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 overload 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 overload response, the control always executes a machining stop and locks the momentary tool.

Example

The entries in columns **AFC-OVLD1** and **AFC-OVLD2** are added to the feedback-control reference power **AFC-LOAD**.

Further information: "AFC teach-in cut", Page 890

Example input for tool wear and tool load monitoring:

Column	Input
AFC-LOAD	30%
AFC-OVLD1	5%
AFC-OVLD2	10%

In this example, the control adds the 5% and 10% to the 30% in each case.

As soon as a value is defined in column **AFC-OVLD1**, the tool will monitor tool wear. When the control used in the example reaches a spindle power of 35% in total, it executes the defined reaction.

20.2 Active Chatter Control (ACC) (#145 / #2-30-1)

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 1684

Requirements

- Active Chatter Control software option (ACC) (#145 / #2-30-1)
- 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** toggle switch in the **Program Run** operating mode or the **MDI** application.

Further information: "The Program Run operating mode", Page 1638

Further information: "The MDI Application ", Page 1221

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

Further information: "The Positions workspace", Page 149

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.
- The Machine Vibration Control software option (MVC) (#146 / #2-24-1) allows influencing the result even more positively.

20.3 Functions for controlling program run

20.3.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION S-PULSE	Program pulsing spindle speed	Page 895
FUNCTION DWELL	Program singular dwell time	Page 896
FUNCTION FEED DWELL	Program cyclic dwell time	Page 897

20.3.2 Pulsing spindle speed with FUNCTION S-PULSE

Application

Using the **FUNCTION S-PULSE** function, you can program a pulsing spindle speed to avoid natural oscillations of the machine, for example.

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 nominal 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 in effect.. 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: "The Positions workspace", Page 149

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 in effect Only if **P-TIME** has been selected Optional syntax element
TO-SPEED	Upper speed limit up to which the pulsing spindle speed will be in effect 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 **FUNCTION S-PULSE** falls below the maximum speed once more.

20.3.3 Programmed dwell time with FUNCTION DWELL

Application

The **FUNCTION DWELL** function allows 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 898

■ Program recurring dwell time

Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 897

Description of function

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

20.3.4 Cyclic dwell time with FUNCTION FEED DWELL

Application

FUNCTION FEED DWELL allows you to program a cyclic dwell time in seconds, such as for forcing chip breaking.

Related topics

- Program a one-time dwell time

  Further information: "Programmed dwell time with FUNCTION DWELL", Page 896

Description of function

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motions.

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, dwell for 0.5 seconds

To navigate to this function:

Insert NC function ▶ Special functions ▶ Functions ▶ FUNCTION FEED ▶ FUNCTION FEED DWELL

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**.

20.4 Cycles with control function

20.4.1 Cycle 9 DWELL TIME

ISO programming
G4

Application

> 🛈 This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
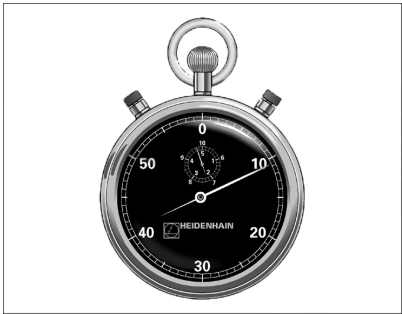
Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle takes effect as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Related topics

■ Dwell time with **FUNCTION FEED DWELL**

Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 897

■ Dwell time with **FUNCTION DWELL**

Further information: "Programmed dwell time with FUNCTION DWELL", Page 896

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

Example

89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5

20.4.2 Cycle 13 ORIENTATION

ISO programming

G36

Application

> ⚙ Refer to your machine manual.
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
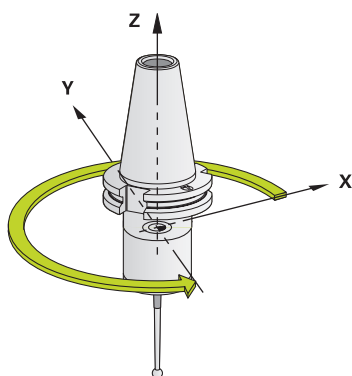
The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

■ Tool changing systems with a defined tool change position

■ Orientation of the transceiver window of HEIDENHAIN 3D 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 manufacturer.

Notes

■ This cycle can be executed in the **FUNCTION MODE MILL** machining mode.

■ 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

20.4.3 Cycle 32 TOLERANCE

ISO programming

G62

Application

> ⚙ Refer to your machine manual.
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
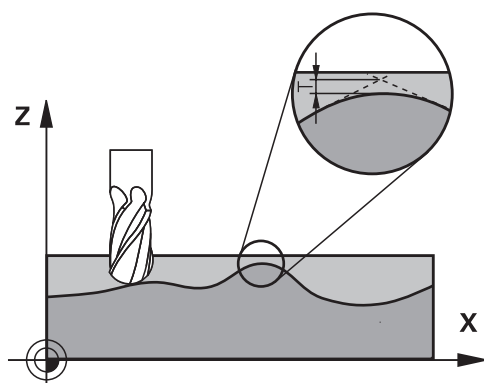
With the entries in Cycle **32** you can influence the result of HSC machining with respect to accuracy, surface definition and speed, in as much as the control has been adapted to the machine's characteristics.

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 manufacturer has implemented these features.

> ℹ 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.

Reset

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
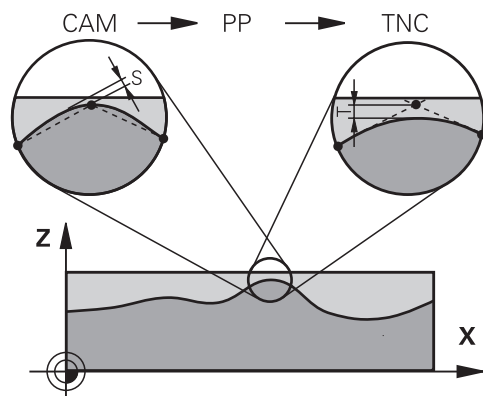
The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The chord error defines the maximum point spacing of NC programs generated in a postprocessor (PP). If the chord error is less than or equal to the tolerance value **T** defined in Cycle **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% L of the CAM chord error.

Related topics

- Working with CAM-generated NC programs
  Further information: "CAM-generated NC programs", Page 947

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **32** is DEF-active which means that it takes effect 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.
- As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine manufacturer).
- 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
	T Tolerance of contour deviation Permitted contour deviation in mm or inch **>0**: The control uses the maximum permitted deviation you have specified. **0**: The control uses a value configured by the machine manufacturer. When skipping this parameter with **NO ENT**, the control uses 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**
	TA Tolerance 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 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**: The control uses the maximum permitted deviation you have programmed. **0**: The control uses a value configured by the machine manufacturer. When skipping this parameter with **NO ENT**, the control uses a value configured by the machine manufacturer. Input: **0...10**

Example

11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.02
13 CYCL DEF 32.2 HSC-MODE:1 TA5

21

Monitoring

21.1 Component monitoring with MONITORING HEATMAP (#155 / #5-02-1)

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 color-coded heatmap on the workpiece.

Related topics

- The **MON** tab of the **Status** workspace
  Further information: "The MON tab (#155 / #5-02-1)", Page 162
- Cycle **238 MEASURE MACHINE STATUS** (#155 / #5-02-1)
  Further information: "Cycle 238 MEASURE MACHINE STATUS (#155 / #5-02-1)", Page 910
- Color the workpiece as a heat map in the simulation
  Further information: "The Workpiece options column", Page 1202

Requirements

- Component monitoring software option (#155 / #5-02-1)
- 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.

The heatmap displays a color image consisting of the following basic colors:

- Green: component works under conditions defined as safe
- Yellow: component works under warning zone conditions
- Red: Overload condition

In addition, the control displays the following colors:

- Light gray: no component was configured
- Dark gray: component cannot be monitored (e.g., due to incorrect or missing details within the configuration)

> **i** Refer to your machine manual.
> The machine manufacturer configures the components.

The control shows these statuses on the workpiece in the simulation and can overwrite the statuses upon subsequent operations.
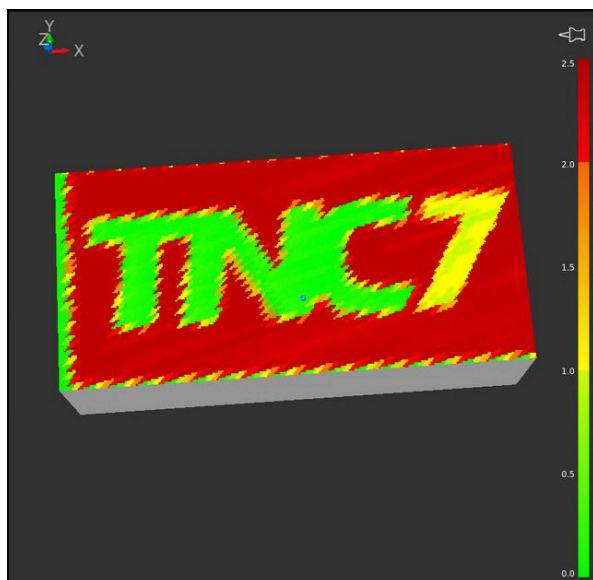
Representation of the component heat map in the simulation with missing pre-machining

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

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **MONITORING** ▶ **MONITORING HEATMAP**

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
File or **QS**	Component to be monitored
	Fixed or variable name
	Selection by means of a selection window
	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 tool breakage). The control shows the change with a slight time delay.

21.2   Cycles for monitoring

21.2.1   Cycle 238 MEASURE MACHINE STATUS (#155 / #5-02-1)

ISO programming

G238

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.

With software option **Component Monitoring** (#155 / #5-02-1) 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 manufacturer. He can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

The machine manufacturer can define warning and error thresholds for the measured values and optionally specify error reactions.

Related topics

■ Component monitoring with **MONITORING HEATMAP** (#155 / #5-02-1)

Further information: "Component monitoring with MONITORING HEATMAP (#155 / #5-02-1)", Page 908

Cycle run

> ℹ️ 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 manufacturer 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: "The MON tab (#155 / #5-02-1)", Page 162

> ⚙️ Your machine manufacturer defines in detail how the axes will move.

Notes

> ⚙️ Cycle **238 MEASURE MACHINE STATUS** can be hidden with the optional machine parameter **hideCoMo** (no. 128904).

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** machining mode.

■ 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
```

21.2.2 Cycle 239 ASCERTAIN THE LOAD (#143 / #2-22-1)

ISO programming

G239

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
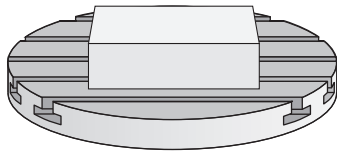
The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With software option **Load Adaptive Control** (#143 / #2-22-1) and Cycle **239 ASCERTAIN THE LOAD**, the control is able to ascertain and adapt the current mass inertia of the load, the current friction forces and the maximum axis acceleration automatically or to reset 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 manufacturer 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 manufacturer.

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 run

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 manufacturer.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the ascertained parameters.

> ℹ 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!**
>
> This cycle may perform extensive movements in one or more axes at rapid traverse! There is a danger of collision!
>
> ▶ Contact your machine manufacturer 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 manufacturer 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** machining mode.
- Cycle **239** takes effect 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
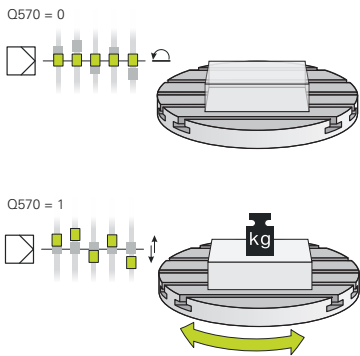
Help graphic	Parameter
Q570 = 0 Q570 = 1	**Q570 Load (0 = Delete/1 = Ascertain)?** Define whether the control will perform a LAC (Load Adaptive Control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters will be reset: **0**: Reset LAC; the values 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

22

Multiple-Axis Machining

22.1 Cycles for cylinder surface machining

22.1.1 Cycle 27 CYLINDER SURFACE (#8 / #1-01-1)

ISO programming
G127

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
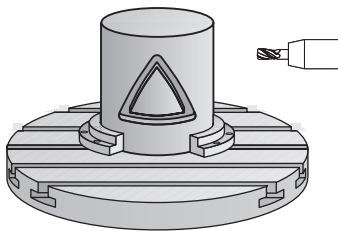
This cycle enables you to program a contour in two dimensions and then transfer it onto a cylindrical surface. Use Cycle **28** to mill guide slots on the cylinder.

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 coordinate data of the unrolled cylinder surface (X coordinates), which define the position of the rotary table, can be entered as desired either in degrees or in mm (or inches) (**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.

> ℹ 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.

> **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.

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

### 22.1.2 Cycle 28 CYLINDRICAL SURFACE SLOT (#8 / #1-01-1)

ISO programming
G128

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
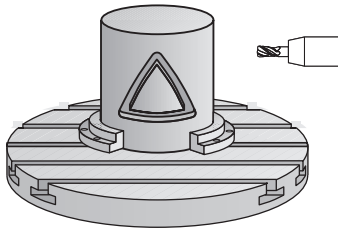
With this cycle you can program a guide slot in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle **27**, with this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel 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 run

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 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. There is a danger of collision!

▶ 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 absolute coordinates (no 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
Y (Z) ... X (C)	**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**
	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

Help graphic	Parameter
	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

22.1.3 Cycle 29 CYL SURFACE RIDGE (#8 / #1-01-1)

**ISO programming**
G129

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
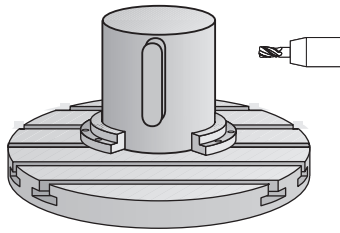
This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle, the control adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center path of the ridge together with the tool radius 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
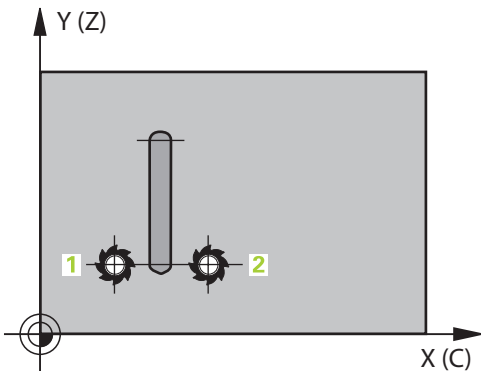
1 The control positions the tool above the starting point of machining. The control calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation 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.

ℹ️ 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.
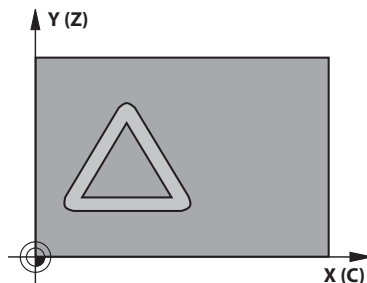
Notes on programming

- 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

22.1.4 Cycle 39 CYL. SURFACE CONTOUR (#8 / #1-01-1)

ISO programming
G139

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
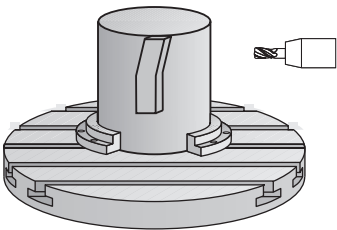
This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel 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.

> **i**   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.

> **i**   ■ 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

22.1.5 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
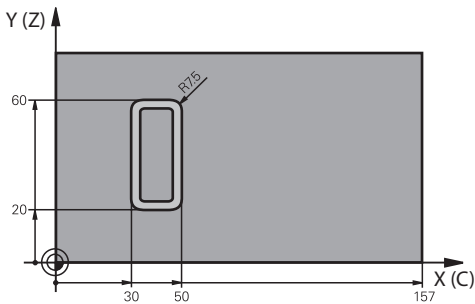
0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7 **;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
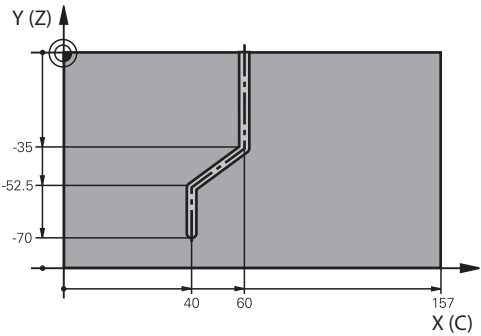
0 BEGIN PGM 4 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call, tool axis (Z), diameter (7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 28 CYLINDRICAL 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	

22.2 Working with the parallel axes U, V and W

22.2.1 Fundamentals

In addition to the main axes X, Y, and Z, the parallel axes U, V, and W, are available. A parallel axis is, for example, a spindle sleeve for boring so that smaller masses are moved on large machines.

Further information: "Programmable axes", Page 194

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 933

- **FUNCTION PARAXMODE**: Select three linear axes for machining
  Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 938

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**. Since the control then continuously offsets the parallel axis, you can for example probe a workpiece even with any position of the W axis.

In this case, the control displays a symbol in the **Positions** workspace.

Further information: "The Positions workspace", Page 149

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.

22.2.2 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: "The Positions workspace", Page 149

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 main axis.

For example, if a parallel-axis movement is performed in the negative W-axis direction, the main 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

Notes

- The **PARAXCOMP MOVE** function can be used only in connection with straight-line blocks (**L**).
- The control allows the use of one active **PARAXCOMP** function per axis only. If you define an axis both in **PARAXCOMP DISPLAY** and in **PARAXCOMP MOVE**, the last executed function will be active.
- Using offset values, you can define a parallel axis shift for the NC program (e.g., in the **W** axis). This allows machining of workpieces with different heights using the same NC program, for example.
  Further information: "Example", Page 936

Notes about machine parameters

The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION PARAXCOMP**, the machine parameter applies to the parallel axes (**U_OFFS**, **V_OFFS**, and **W_OFFS**) only. If there are no offsets, the control behaves as described in the functional description.

Further information: "Description of function", Page 934

Further information: "Basic transformation and offset", Page 1714

- If the machine parameter has not been defined for the parallel axis or has been defined with **FALSE**, the offset is only active in the parallel axis. The preset of the programmed parallel-axis coordinates is shifted by the offset value. The coordinates of the main axis still reference the workpiece preset.
- If the machine parameter for the parallel axis has been defined with **TRUE**, the offset will be active in the parallel and main axes. The presets of the programmed parallel and main axis coordinates are shifted by the offset value.

Example

This example shows the effect of the optional machine parameter **presetToAlignAxis** (no. 300203)

Machining is done on a gantry-type milling machine using a spindle sleeve as the **W** axis (parallel to the main **Z** axis). The **W_OFFS** column of the preset table contains the value **-10**. The Z value of the workpiece preset is located at the machine datum.

Further information: "Presets in the machine", Page 196

11 L Z+100 W+0 R0 FMAX M91	; Position the **Z** and **W** axes in the machine coordinate system **M-CS**
12 FUNCTION PARAX COMP DISPLAY W	; Activate the sum display
13 L Z+0 F1500	; Position the Z axis at 0
14 L W-20	; Move the W axis to working depth

In the first NC block, the control positions the **Z** and **W** axes relative to the machine datum, i.e. independent of the workpiece preset. In the **RFACTL** mode, the position display indicates the values **Z+100** and **W+0**. In the **ACTL.** mode, the control takes **W_OFFS** into account and displays the values **Z+100** and **W+10**.

Further information: "Position displays", Page 172

In NC block **12**, the control activates sum display for the **ACTL.** and **NOML.** modes of the position display. The control displays the movements of the W axis in the position display of the Z axis.

The result depends on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
The control takes the offset into account in the W axis only. The value of the Z axis display remains unchanged.	The control takes the offset into account in the **W** and **Z** axes. The **ACTL.** display of the Z axis is changed by the offset value.
Position-display values: ■ **RFACTL** mode: **Z+100**, **W+0** ■ **ACTL.** mode: **Z+100**, **W+10**	Position-display values: ■ **RFACTL** mode: **Z+100**, **W+0** ■ **ACTL.** mode: **Z+110**, **W+10**

In NC block **13**, the control moves the Z axis to the programmed coordinate **0**.

The result depends on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
The control moves the Z axis by 100 mm.	The coordinates of the Z axis reference the offset. To reach the programmed coordinate **0**, the axis must move by 110 mm.
Position-display values: ■ **RFACTL** mode: **Z+0**, **W+0** ■ **ACTL.** mode: **Z+0**, **W+10**	Position-display values: ■ **RFACTL** mode: **Z-10**, **W+0** ■ **ACTL.** mode: **Z+0**, **W+10**

In NC block **14**, the control moves the W axis to the programmed coordinate **-20**. The coordinates of the W axis reference the offset. To reach the programmed coordinate, the axis must move by 30 mm. Since the sum display has been activated, the control displays the movement in the **ACTL.** display of the Z axis as well.

The values in the position display depend on the setting of the **presetToAlignAxis** machine parameter:

FALSE or not defined	TRUE
Position-display values:	Position-display values:
■ **RFACTL** mode: **Z+0**, **W-30**	■ **RFACTL** mode: **Z-10**, **W-30**
■ **ACTL.** mode: **Z-30**, **W-20**	■ **ACTL.** mode: **Z-30**, **W-20**
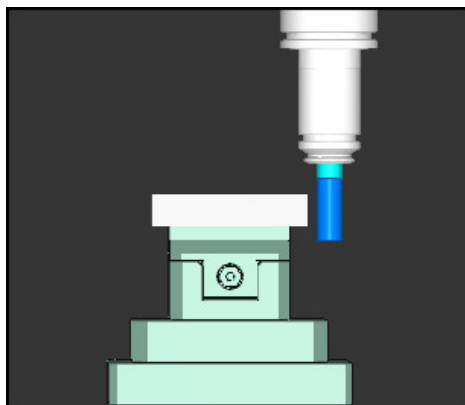
The tool tip is lower by the offset value than programmed in the NC program (**RFACTL W-30** instead of **W-20**).
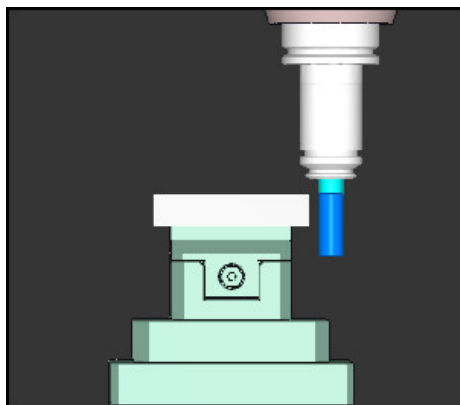
The tool tip is lower by the twice the offset value than programmed in the NC program (**RFACTL Z-10**, **W-30** instead of **Z+0**, **W-20**).

> **i** If you only move the W axis while the **PARAXCOMP DISPLAY** function is active, the control takes the offset into account only once, independent of the setting of the **presetToAlignAxis** machine parameter.

22.2.3 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 traverses and contour descriptions in the main 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 933

Description of function

If the **PARAXMODE** function is active, the control uses the axes defined in the function to execute the programmed traverses. If the control is to move the main axis deselected by **PARAXMODE**, you can identify this axis by additionally entering the **&** character. The **&** character then refers to the main axis.

Further information: "Moving the main axis and the parallel axis", Page 939

Define three axes with the **PARAXMODE** function (e.g., **FUNCTION PARAXMODE X Y W**) to be used by the control for programmed traverses.

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: "The Positions workspace", Page 149

FUNCTION PARAXMODE OFF

Use the **PARAXMODE OFF** function to deactivate the parallel-axis function. The control then uses the main axes defined by the machine manufacturer.

The control resets the **PARAXMODE ON** parallel-axis function via the following functions:

- Selection of an 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 main 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 300

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.

■ In order for the control to offset the main axis deselected with **PARAXMODE**, enable the **PARAXCOMP** function for this axis.

■ Additional positioning of a main 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 172

Notes about machine parameters

■ In the machine parameter **noParaxMode** (no. 105413), you define whether the control provides the functions **PARAXCOMP** and **PARAXMOVE**.

■ 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).

  ■ If the machine parameter has not been defined for the main axis or has been defined with **FALSE**, the offset only applies to the axis programmed with **&**. The coordinates of the parallel axis still reference the workpiece preset. Despite the offset, the parallel axis will move to the programmed coordinates.

  ■ If the machine parameter for the main axis has been defined with **TRUE**, the offset applies to the main axis and the parallel axis. The presets of the main and parallel axis coordinates are shifted by the offset value.

22.2.4 Parallel axes in conjunction with machining cycles

You can also use most machining cycles of the control with parallel axes.

Further information: "Working with cycles", Page 222

Touch-probe cycles (#17 / #1-05-1) cannot be used in conjunction with parallel axes.

22.2.5 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	

22.3 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.

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 933

Description of function
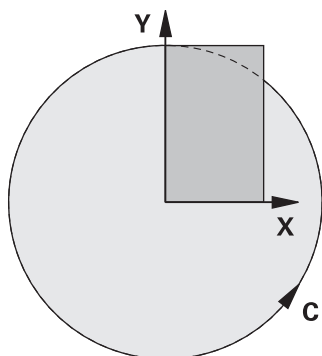
When the polar kinematics are active, the control displays an icon in the **Positions** workspace. This icon covers the icon for the **PARAXCOMP DISPLAY** function.

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 at the table opposite to the selected linear axes can be used as the rotary axis.
- The two selected linear axes thus span a plane that also includes the rotary axis.
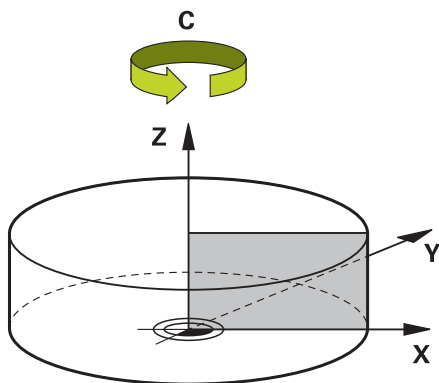
The following scenarios lead to deactivation of the polar kinematics:

- 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 was positioned when the function was activated.
	If the radial axis is positioned at 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 was positioned when the function was 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

> ℹ 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 943 Only when **AXES** is selected
POLE:	Selection of machining in the pole **Further information:** "POLE options", Page 943 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 feed-rate variations may occur in this area. For this reason, ideally use the following **POLE** option: **SKIPPED**.
- Polar kinematics cannot be combined with the following functions:
  - Traverses with **M91**

    Further information: "Traversing in the machine coordinate system M-CS with M91", Page 967
  - Tilting the working plane (#8 / #1-01-1)
  - **FUNCTION TCPM** or **M128** (#9 / #4-01-1)
- Note that the traversing range of the axes may be limited.

  Further information: "Notes on software limit switches for modulo axes", Page 957

  Further information: "Traverse Limits", Page 1782

Notes about machine parameters

- The machine manufacturer uses the optional machine parameter **kindOfPref** (no. 202301) to define the behavior of the control when the path of the tool center point passes through the polar axis.

- The machine manufacturer uses the optional machine parameter **preset-ToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION POLARKIN**, the machine parameter applies only to the rotary axis that rotates about the tool axis (in most cases **C_OFFS**).

  Further information: "Comparison of offset and 3D basic rotation", Page 1288

  - If the machine parameter axis has not been defined or has been set to **TRUE**, the offset can be used to compensate a misalignment of the workpiece in the plane. The offset affects the orientation of the workpiece coordinate system **W-CS**.

    Further information: "Workpiece coordinate system W-CS", Page 697

  - If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

22.3.1 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	

22.4 CAM-generated NC programs

Application

CAM-generated NC programs are created externally of the control using CAM systems.

CAM systems provide a comfortable and sometimes unique solution in connection with 4-axis simultaneous machining.
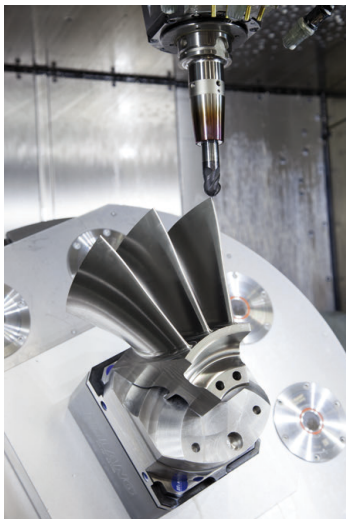
For CAM-generated NC programs to be able to use the full performance potential of the control and to provide you with such options as intervention and correction, certain requirements must be met.

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 952

The process chain specifies the path from a design to the finished workpiece.
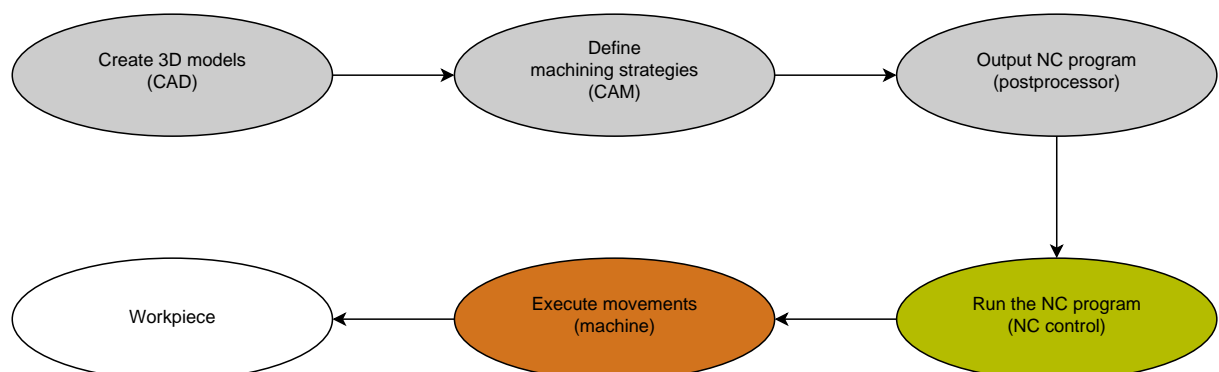
Related topics

- Using 3D data directly at the control
  Further information: "Opening CAD files with CAD Viewer", Page 1107
- Programming graphically
  Further information: "Graphical programming", Page 1089

22.4.1 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 four axes, without **M128** or **FUNCTION TCPM**
- Output with up to four axes, with **M128** or **FUNCTION TCPM** (#9 / #4-01-1)

> **i** Prerequisites for 4-axis machining:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1) for **M128** or **FUNCTION TCPM**

> **i** The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.
> If the axis position does not change, you can nevertheless program more than four axes.

If the machine kinematics and the exact tool data are available to the CAM system, you can output 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 (#9 / #4-01-1)", Page 787

Further information: "Presets on the tool", Page 253

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 C+45 R0 F5000	; 4-axis without **M128**

11 L X+88 Y+23.5375 Z-8.3 C+45 R0 F5000 M128	; 4-axis with **M128**

Output with vectors
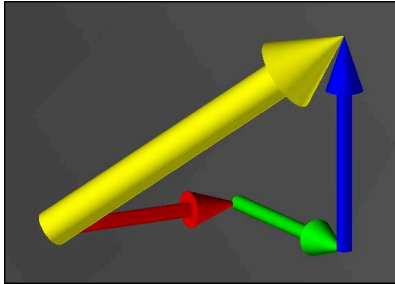
From the point of view of physics and geometry, a vector is a directed variable that describes a direction and a length.

When outputting with vectors, the control requires at least one vector that specifies the direction of the surface normal or the tool angle of inclination. Optionally, the NC block contains both vectors.

> **ⓘ** Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1)

> **ⓘ** The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.
>
> If the axis position does not change, you can nevertheless program more than four axes.

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 TY-0.8764339 TZ+0.2590319 M128	; 4-axis with M128, surface normal vector and tool orientation

Structure of an NC block with vectors
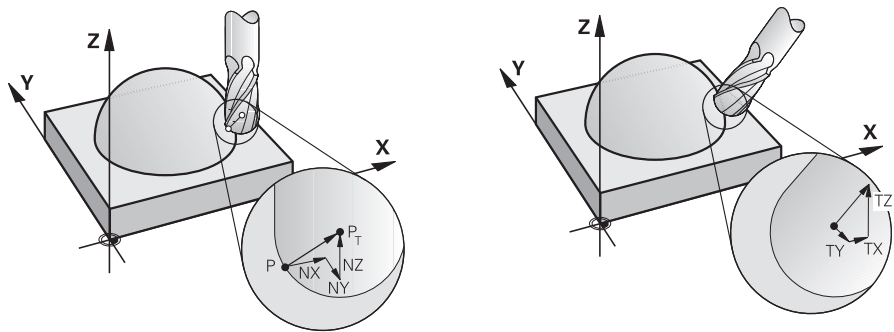
Surface normal vector perpendicular to the contour

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 Optional syntax element
TX TY TZ	Components of the tool direction vector Optional syntax element

22.4.2   Types of machining according to number of axes

3-axis machining
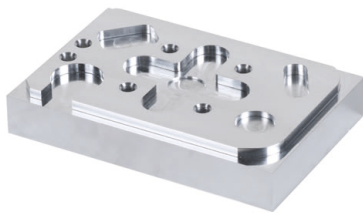
If only the linear axes **X**, **Y** and **Z** are required for machining a workpiece, 3-axis machining takes place.

3+2-axis machining

If tilting of the working plane is required for machining a workpiece,
3+2-axis machining takes place.

> ℹ️ Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)

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 for
the offset that is created in the linear axes.

Inclined machining is used in conjunction with undercuts and short tool clamping
lengths.

> ℹ️ Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1)

4-axis machining

In 4-axis machining, also referred to as 4-axis simultaneous machining, the machine moves four axes at the same time.

> **ℹ** Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1)

22.4.3 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.

> **ℹ** 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 required in conjunction with collision testing (#40 / #5-03-1).
> - 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 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 errors and Cycle 32 TOLERANCE
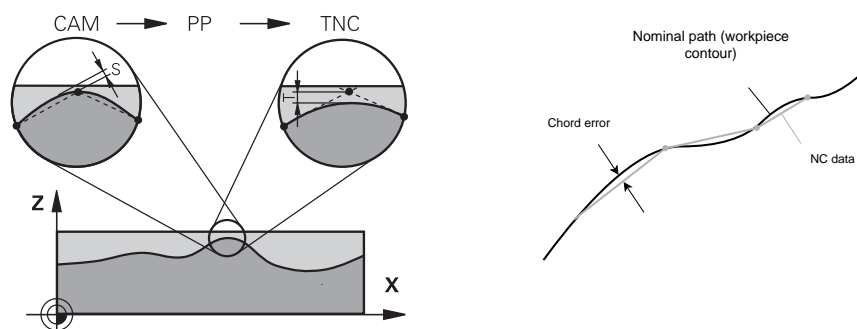
- In roughing, the focus is on the processing speed.
  The sum of the chord error and the tolerance **T** in Cycle **32 TOLERANCE** must be smaller than the contour allowance, otherwise contour violations may occur.

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 902

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**).

- If a subprogram call and a subprogram definition are separated by multiple NC blocks, program execution might be interrupted due to the calculation effort. Use the following options to avoid problems such as dwell marks due to interruptions:

  - Put subprograms that define retraction positions at the beginning of the program. Thus, the control "knows" where to find the subprogram when it is called later.

  - Use a separate NC program for machining positions or coordinate transformations. This ensures that the control simply needs to call that program when safety positions and coordinate transformations are required in the NC program.

- 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.
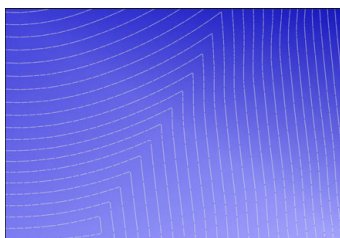
Tool paths with strong changes in direction at transitions
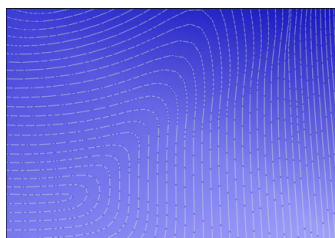
Tool paths with rounded transitions

- 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.

- Avoid incorrect positioning by outputting the **PLANE** functions (#8 / #1-01-1) with **MOVE** or **TURN** without using separate positioning blocks. If you output **STAY** and position the rotary axes separately, use the variables **Q120** to **Q122** instead of fixed-axis values.

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737

- 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.

- When the machine moves multiple axes at the same time, kinematic errors of the axes might sum up. Move as few axes as possible simultaneously.
- Avoid unnecessary feed-rate limitations, which you can define for compensation movements within **M128** or the function **FUNCTION TCPM** (#9 / #4-01-1).
  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
- Take into account the machine-specific behavior of rotary axes.
  Further information: "Notes on software limit switches for modulo axes", Page 957

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 253

Notes on 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 1006
- Provide a better overview by structuring the NC programs. One method is to use subprograms within the NC programs. If possible, divide larger projects into multiple separate NC programs.
  Further information: "Programming Techniques", Page 359
- Support correction options by outputting contours with tool radius correction.
  Further information: "Tool radius compensation", Page 798
- Use structure items to enable fast navigation within the NC programs.
  Further information: "Structuring of NC programs", Page 1166
- Use comments to communicate important information about the NC program.
  Further information: "Adding comments", Page 1164

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: "The Simulation Workspace", Page 1197

■ Take into account the machine-specific behavior of rotary axes.
Further information: "Notes on software limit switches for modulo axes", Page 957

■ Make sure that the required tools are available and that the remaining service life is sufficient.
Further information: "Tool usage test", Page 287

■ 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 902

> ⚙ 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 vectors, it is possible to correct tool movements in three dimensions.
Further information: "Output formats of NC programs", Page 948

■ Software options enable further optimizations.
Further information: "Functions and function packages", Page 959
Further information: "Software options", Page 78

Notes on software limit switches for modulo axes

> **i** The following information on software limit switches for modulo axes also applies to traversing limits.
> **Further information:** "Traverse Limits", Page 1782

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° (such as 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° (such as 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°.

22.4.4 Functions and function packages

ADP motion control
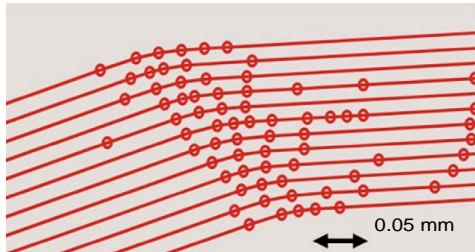
Distribution of points
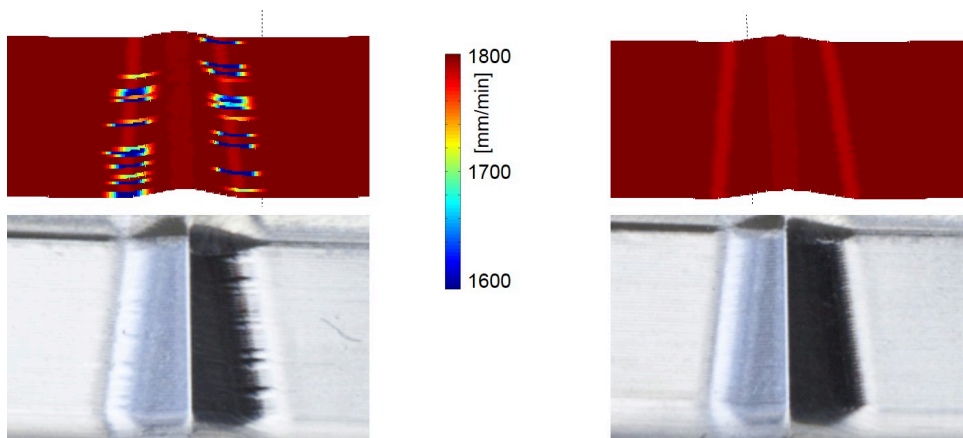
Comparison without and with ADP

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
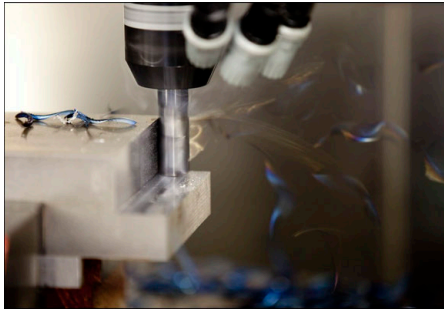
The Dynamic Efficiency package of functions enables you to increase process reliability in heavy machining and roughing in order to improve efficiency.

Dynamic Efficiency includes the following software features:

- Active Chatter Control (ACC (#45 / #2-31-1))
- Adaptive Feed Control (AFC (#45 / #2-31-1))
- Trochoidal milling cycles (#167 / #1-02-1)

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
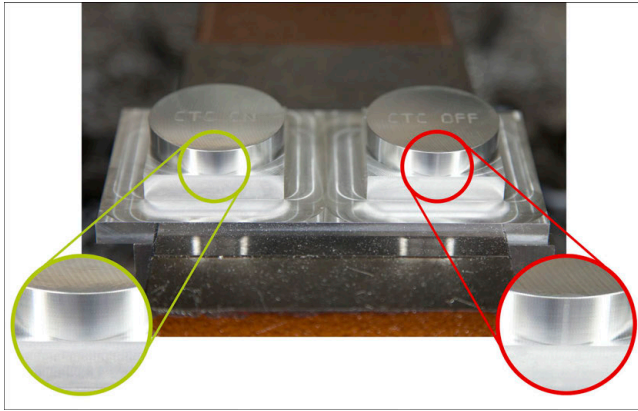
The Dynamic Precision package of functions enables you to machine quickly and accurately, and with high surface quality.

Dynamic Precision includes the following software functions:

- Cross Talk Compensation (CTC (#141 / #2-20-1))
- Position Adaptive Control (PAC (#142 / #2-21-1))
- Load Adaptive Control (LAC (#143 / #2-22-1))
- Motion Adaptive Control (MAC (#144 / #2-23-1))
- Machine Vibration Control (MVC (#146 / #2-24-1))

The functions each provide decisive improvements. They can be combined and also mutually complement each other:

- CTC increases the accuracy in the acceleration phases.
- MVC allows to machine better surfaces.
- CTC and MVC 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.**

23

Miscellaneous Functions

23.1 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: "The Manual operation application", Page 186

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.

23.1.1 Programming the STOP function

To program the **STOP** function:

| STOP | ▶ | Select **STOP** |
| | > | The control creates a new NC block with the **STOP** function. |

23.2 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 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	■	
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	□	
M30 Function is Identical to **M2**	■	
M89 Call the cycle modally	□ ■	Page 227
M91	□	Page 967

Function	Effect	Further information
Traverse in the machine coordinate system **M-CS**		
M92 Traverse in the **M92** coordinate system	□	Page 968
M94 Reduce the display for rotary axes to under 360°	□	Page 970
M97 Machine small contour steps	■	Page 972
M98 Machine open contours completely	■	Page 974
M99 Call a cycle once per block	■	Page 227
M101 Automatically insert a replacement tool	□	Page 999
M102 Reset **M101**	■	
M103 Reduce feed rate for infeed movements	□	Page 975
M107 Permit positive tool oversizes	□	Page 1001
M108 Check the radius of the replacement tool Reset **M107**	■	Page 1003
M109 Adapt feed rate for circular paths	□	Page 976
M110 Reduce feed rate for inner radii	□	
M111 Reset **M109** and **M110**	■	
M116 Interpret feed rate for rotary axes as mm/min	□	Page 978
M117 Reset **M116**	■	
M118 Activate handwheel superimpositioning	□	Page 979
M120 Pre-calculate the radius-compensated contour (look ahead)	□	Page 981
M126 Shorter-path traverse of rotary axes	□	Page 985
M127 Reset **M126**	■	
M128	□	Page 986

Function	Effect	Further information
Automatically compensate for tool inclination (TCPM)		
M129 Reset **M128**	■	
M130 Traverse in the non-tilted input coordinate system **I-CS**	□	Page 969
M136 Interpret feed rate as mm/rev	□	Page 991
M137 Reset **M136**	■	
M138 Take rotary axes into account during machining operations	□	Page 992
M140 Retract in the tool axis	□	Page 993
M141 Suppress touch probe monitoring	□	Page 1004
M143 Rescind basic rotations	□	Page 995
M144 Factor the tool offset into the calculations	□	Page 995
M145 Reset **M144**	■	
M148 Automatically lift off upon an NC stop or a power failure	□	Page 996
M149 Reset **M148**	■	
M197 Prevent rounding off of outside corners	■	Page 997

23.3 Miscellaneous functions for coordinate entries

23.3.1 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 in effect in the machine coordinate system **M-CS**.

Further information: "Machine coordinate system M-CS", Page 692

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 196

> ⚙ 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 798
- The control uses the tool carrier reference point when positioning in the tool axis.
  Further information: "Presets in the machine", Page 196
- 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 172
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset to the simulation. In this constellation you can simulate traverse movements with **M91**.
  Further information: "The Visualization options column", Page 1200
- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

23.3.2 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 in effect in the **M92** coordinate system.

Further information: "Presets in the machine", Page 196

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 196

> ⚙ 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 798
- The control uses the tool carrier reference point when positioning in the tool axis.
  Further information: "Presets in the machine", Page 196
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset to the simulation. In this constellation you can simulate traverse movements with **M92**.
  Further information: "The Visualization options column", Page 1200
- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

23.3.3 Traversing in the non-tilted input coordinate system I-CS with M130

Application

Coordinates of a straight line entered with **M130** are in effect 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 300

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 702

Notes

NOTICE

Danger of collision!

The miscellaneous function **M130** is in effect 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.

23.4 Miscellaneous functions for path behavior

23.4.1 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: "The Positions workspace", Page 149

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 using the shortest traverse path. If the traverse paths in both directions are identical, you can pre-position the rotary axis and thus also influence the direction of rotation. Within the **PLANE** functions, you can also select a tilting solution.

  Further information: "Tilting solution", Page 774
- 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°.

23.4.2 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.

> **i** HEIDENHAIN recommends using the more powerful function **M120** (#21 / #4-02-1) instead of **M97**.
>
> 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** (#21 / #4-02-1)
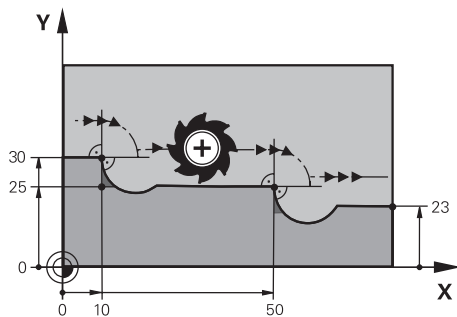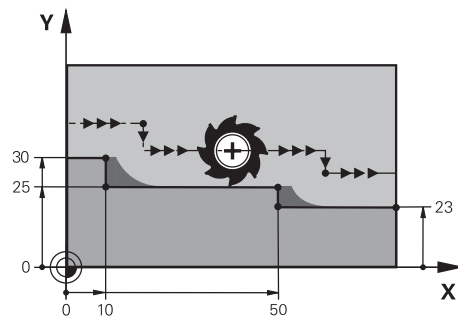
  Further information: "Pre-calculating a radius-compensated contour with M120 (#21 / #4-02-1)", Page 981

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.

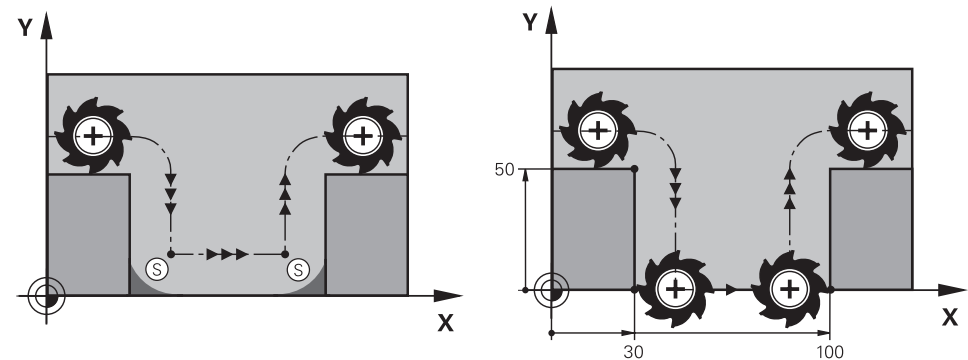### 23.4.3 Machining open contour corners with M98

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.

23.4.4 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 in effect with an active tilted working plane coordinate system **WPL-CS**. The feed rate reduction is then active during infeed movements in the virtual tool axis **VT**.

23.4.5 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 active for circular paths within these machining cycles.

23.4.6 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 active for circular paths within these machining cycles.

23.4.7 Interpreting the feed rate for rotary axes in mm/min with M116 (#8 / #1-01-1)

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 Advanced Functions Set 1 (#8 / #1-01-1)

Description of function

Effect

M116 is active 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. (#8 / #1-01-1)

  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736

- It is not possible to combine **M116** with **M128** or **FUNCTION TCPM** (#9 / #4-01-1). 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 992

- Without **M128** or **FUNCTION TCPM** (#9 / #4-01-1), **M116** can be in effect for multiple rotary axes at the same time.

23.4.8 Activating handwheel superimpositioning with M118 (#21 / #4-02-1)

Application

With **M118** the control activates handwheel superimpositioning. You can then perform manual corrections by handwheel during program run.

Requirements

- Handwheel
- Software option Advanced Functions Set 3 (#21 / #4-02-1)

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 in effect in the machine coordinate system **M-CS**.
- 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.
- Handwheel superimpositioning with **M118** in combination with Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) is possible only at a standstill.

  In order to use **M118** without restrictions, either deactivate **DCM** (#40 / #5-03-1) or activate a kinematics model without collision objects.

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848
- Handwheel superimpositioning is also effective in the **MDI** application.

  Further information: "The MDI Application ", Page 1221
- If you want to use **M118** with clamped axes, you must unclamp them first.

23.4.9 Pre-calculating a radius-compensated contour with M120 (#21 / #4-02-1)

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 Advanced Functions Set 3 (#21 / #4-02-1)
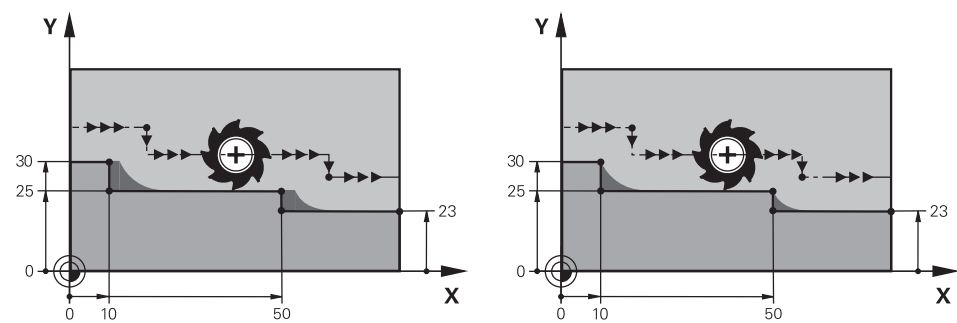
Description of function

Effect

M120 takes effect at the start of the block and remains active beyond the milling cycles.

M120 can be reset by the following NC functions:

- **M120 LA0**
- **M120** without **LA**
- Radius compensation **R0**
- Departure functions (e.g., **DEP LT**)

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 radius compensation is active and you execute the following functions, the control aborts program run and displays an error message:
  - **PLANE** functions (#8 / #1-01-1)
  - **M128** (#9 / #4-01-1)
  - **FUNCTION TCPM** (#9 / #4-01-1)
  - **CALL PGM**
  - Cycle **12 PGM CALL**
  - Cycle **32 TOLERANCE**
  - Cycle **19 WORKING PLANE**

    > **ⓘ** You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Example
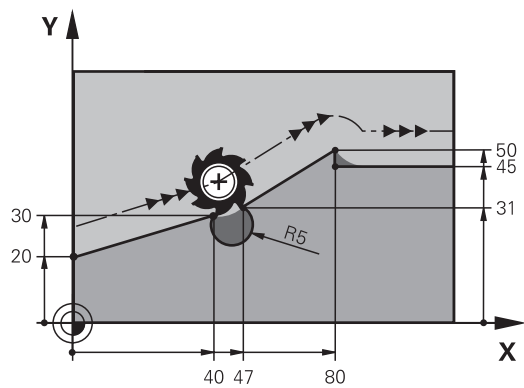
0 BEGIN PGM "M120" MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-10	
2 BLK FORM 0.2 X+110 Y+80 Z+0	; Workpiece blank definition
3 TOOL CALL 6 Z S1000 F1000	; Insert the tool with diameter 12
4 L X-5 Y+26 R0 FMAX M3	; Move in the working plane
5 L Z-5 R0 FMAX	; Infeed in the tool axis
6 L X+0 Y+20 RL F AUTO M120 LA5	; 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

23.4.10 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 affects only 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 using the shortest traverse path. If the traverse paths in both directions are identical, you can pre-position the rotary axis and thus also influence the direction of rotation. Within the **PLANE** functions, you can also select a tilting solution.

  Further information: "Tilting solution", Page 774

- 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°.

23.4.11 Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)

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).

> **ℹ** 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 (#9 / #4-01-1)", Page 787

Requirements

- Machine with rotary axes
- Kinematics description

> **⚙** Refer to your machine manual.
> The machine manufacturer creates the kinematics description of the machine.

- Software option Advanced Functions Set 2 (#9 / #4-01-1)

Description of function

Effect

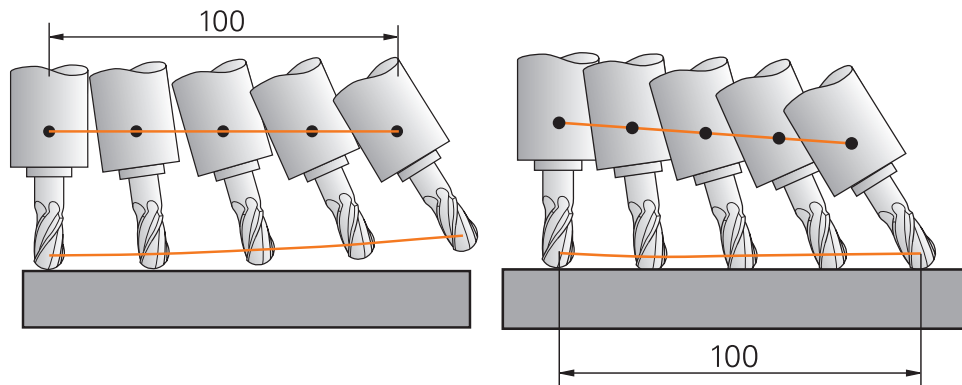**M128** takes effect at the start of the block.

You can reset **M128** with the following functions:

- **M129**
- **FUNCTION RESET TCPM**
- In the **Program Run** operating mode, select a different NC program

> **ℹ** **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.

> 🛈 The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. Please note that the compensation movement is performed in up to three axes.

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 253

> 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 (#9 / #4-01-1)", Page 816

Further information: "Output with vectors", Page 949

- 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** icon in the **Positions** workspace.

  Further information: "The Positions workspace", Page 149
- **M128** and **FUNCTION TCPM** with **AXIS POS** selected do not take into account an active 3D basic rotation. Program **FUNCTION TCPM** with **AXIS SPAT** selected, or CAM outputs with **LN** straight lines and a tool vector.

  Further information: "Basic rotation and 3D basic rotation", Page 707

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
- 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 690
- If you run the following functions while **M128** is active, then the control cancels program run and issues an error message:
  - **M91**
  - **M92**
  - **M144**
  - Calling a tool with **TOOL CALL**
  - Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) and simultaneous use of **M118** (#21 / #4-02-1)

Notes about machine parameters

- 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.
- The machine manufacturer uses the optional machine parameter **preset-ToAlignAxis** (no. 300203) to define for each axis how the control will interpret offset values. With **FUNCTION TCPM** and **M128**, the machine parameter is relevant only for the rotary axis that rotates about the tool axis (mostly **C_OFFS**).

  Further information: "Basic transformation and offset", Page 1714

    - If the machine parameter is not defined or is defined with the value **TRUE**, then you can compensate for a workpiece misalignment in the plane with the offset. The offset affects the orientation of the workpiece coordinate system **W-CS**.

      Further information: "Workpiece coordinate system W-CS", Page 697

    - If the machine parameter is defined with the value **FALSE**, then you cannot compensate for a workpiece misalignment in the plane. The control does not take the offset into account during program run.

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 253

- 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 797

  For Dynamic Collision Monitoring (DCM (#40 / #5-03-1)), you need to define the actual tool length in tool management, too.

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 848

- 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 (#9 / #4-01-1)", Page 787

Definition

Abbreviation	Definition
TCPM (tool center point management)	Maintain the position of the tool location point **Further information:** "Presets on the tool", Page 253

23.4.12 Interpreting the feed rate as mm/rev with M136

Application

With **M136**, the control interprets the feed rate as millimeters per spindle revolution. The feed rate depends on the spindle speed.

Description of function

Effect

M136 takes effect at the start of the block.

In order to reset **M136**, program **M137**.

Application example

11 M136	; Switch interpretation of the feed rate to mm/rev

With **M136** the control interprets the feed rate as millimeters per revolution.

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**.
- When you move the axes while **M136** is active, the control will display the feed rate in mm/rev in the **Positions** workspace and on the **POS** tab of the **Status** workspace.
  Further information: "The Positions workspace", Page 149
  Further information: "POS tab", Page 164
- **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.

23.4.13 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** (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 986
- **FUNCTION TCPM** (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
- **PLANE** functions (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 737
- Cycle **19 WORKING PLANE** (#8 / #1-01-1)

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 TURN 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.
- The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.
- 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.

23.4.14 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 Dynamic Collision Monitoring (DCM (#40 / #5-03-1)). 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 in effect 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 703

- 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

23.4.15 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 705

Note

The function **M143** is not permitted with mid-program startup.

Further information: "Block scan for mid-program startup", Page 1649

23.4.16 Taking the tool offset into account in calculations M144 (#9 / #4-01-1)

Application

The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.

> **i** HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM** (#9 / #4-01-1) instead of **M144**.

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787

Requirement

- Software option Advanced Functions Set 2 (#9 / #4-01-1)

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 967
- 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 (#8 / #1-01-1)", Page 737

  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 690

23.4.17 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

> ℹ 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 881

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 269

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 liftoff direction. If you program **M148**, the control will automatically liftoff the tool in the 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 liftoff is active.
- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum liftoff height.
- In machine parameter **feed** (no. 201405), the machine manufacturer defines the speed of liftoff movement.

23.4.18 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.
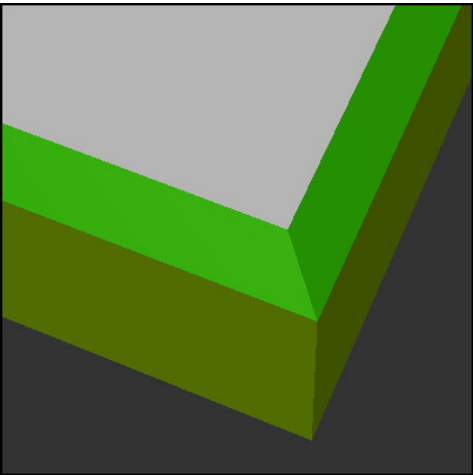
Description of function

Effect

M197 is in effect blockwise and only for radius-compensated outside corners.

Application example
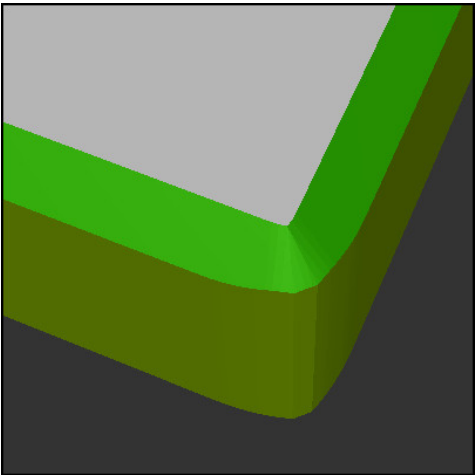
Contour without **M197**

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

23.5 Miscellaneous functions for tools

23.5.1 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

The number of the replacement tool must have been defined 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 269

> **i** 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 1003

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 269
- 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 258
- 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!

Notes on tool change

- The control performs the automatic tool change at a suitable point in the NC program.
- If you do not define a replacement tool in the **RT** column and call the tool via its tool name, the control will switch to a tool with the same name once the maximum tool age **TIME2** has been reached.
  Further information: "Tool name", Page 256
- 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 toler-ance)	Number of NC blocks by which a tool change may be delayed.

23.5.2 Permitting positive tool oversizes with M107 (#9 / #4-01-1)

Application

With **M107** (#9 / #4-01-1), the control does not interrupt machining in case a positive delta value is measured. The function is in effect with active 3D tool compensation and for **LN** straight lines.

Further information: "3D tool compensation (#9 / #4-01-1)", Page 805

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 948

Requirement

■ Software option Advanced Functions Set 2 (#9 / #4-01-1)

Description of function

Effect

M107 takes effect at the start of the block.

In order to reset **M107**, program **M108**.

Application example
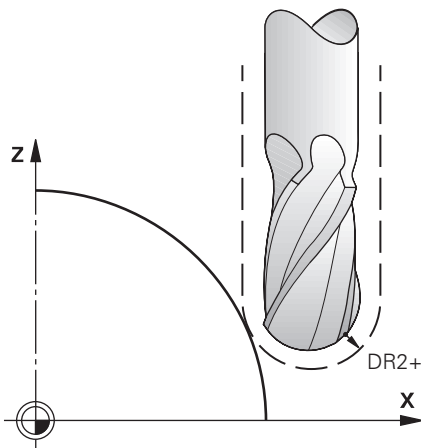
11 TOOL CALL 1 Z S5000 DR2:+0.3	; Insert a tool with a positive delta value
12 M107	; Permit positive delta values

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 (#9 / #4-01-1)", Page 816

■ 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 (#9 / #4-01-1)", Page 809

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

23.5.3 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 999

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** (#9 / #4-01-1).

Further information: "Permitting positive tool oversizes with M107 (#9 / #4-01-1)", Page 1001

23.5.4 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 (#17 / #1-05-1)", Page 1535

Further information: "Cycle 4 MEASURING IN 3-D (#17 / #1-05-1) ", Page 1537

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

---

24

**Variable
Programming**

24.1 Overview of variable programming

The control provides the following options for variable programming in the **FN** folder of the **Insert NC function** window:

Function group	Further information
Basic arithmetic operations	Page 1020
Trigonometric functions	Page 1022
Circle calculations	Page 1024
Jump commands	Page 1026
Special functions	Page 1027
	Page 1039
SQL statements	Page 1066
String functions	Page 1048
Counters	Page 1058
Calculations using formulas	Page 1043
Function for the definition of complex contours	Page 383

24.2 Variables: Q, QL, QR and QS parameters

24.2.1 Basics

Application

You can use the Q, QL, QR and QS parameters of the control, also referred to as variables, to take measurement results into account dynamically within calculations while machining.

For instance, you can program the following syntax elements variably:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

This means that the same NC program can be used for different workpieces and values have to be changed in only one central place.

Description of function
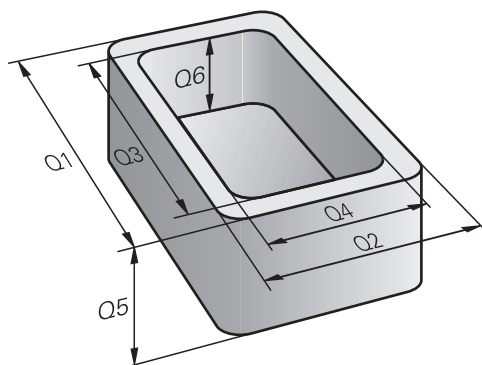
Variables always consist of letters and numbers. The letters determine the type of variable and the numbers its range.

For each variable type, you can define the variable range that the control will display on the **QPARA** tab of the **Status** workspace.

Further information: "Defining the contents of the QPARA tab", Page 175

Variable types

The control provides the following variables for numerical values:

- Q parameters

  Further information: "Q parameters", Page 1008

- QL parameters

  Further information: "QL parameters", Page 1008

- QR parameters

  Further information: "QR parameters", Page 1008

In addition, the control provides QS parameters for alpha-numeric values (e.g., texts).

Further information: "QS parameters", Page 1009

Q parameters

Q parameters affect all NC programs in the control's memory.

Q and QS parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program.

The control provides the following Q parameters:

Variable range	Meaning
0 to 99	User-defined Q parameters, if there are no overlaps with the HEIDENHAIN SL cycles
100 to 199	Q parameters for special functions on the control that can be read by user-defined NC programs or by cycles
200 to 1199	Q parameters for functions defined by HEIDENHAIN (e.g., cycles)
1200 to 1399	Q parameters for functions defined by the machine manufacturer (e.g., cycles)
1400 to 1999	User-defined Q parameters

QL parameters

QL parameters are active locally within an NC program.

The control provides the following QL parameters:

Variable range	Meaning
0 to 499	User-defined QL parameters

QR parameters

QR parameter affect all NC programs in the control's memory; they are retained even after a restart of the control.

The control provides the following QR parameters:

Variable range	Meaning
0 to 99	User-defined QR parameters
100 to 199	QR parameters for functions defined by HEIDENHAIN (e.g., cycles)
200 to 499	QR parameters for functions defined by the machine manufacturer (e.g., cycles)

QS parameters

QS parameters affect all NC programs in the control's memory.

The following characters can be used within QS parameters:

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 ; ! # $ % & ' ( ) + , - . / : < = > ? @ [ ] ^ _ ` *

QS parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program.

The control provides the following QS parameters:

Variable range	Meaning
0 to 99	User-defined QS parameters, if there are no overlaps with the HEIDENHAIN cycles
100 to 199	QS parameters for special functions on the control that can be read by user-defined NC programs or by cycles
200 to 1199	QS parameters for functions defined by HEIDENHAIN (e.g., cycles)
1200 to 1399	QS parameters for functions defined by the machine manufacturer (e.g., cycles)
1400 to 1999	User-defined QS parameters

The Q parameter list window

In the **Q parameter list** window, you can view and edit the values of all variables.
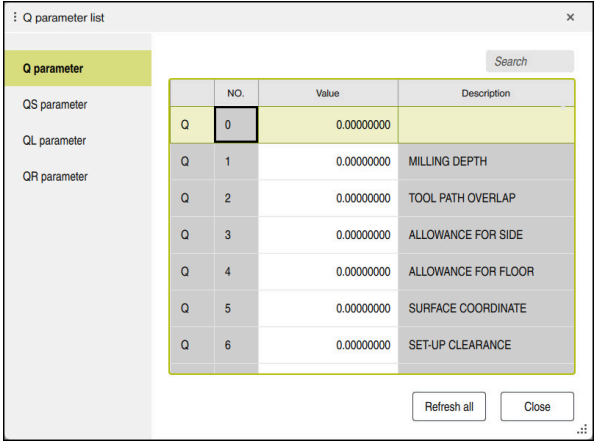
The **Q parameter list** window, showing the Q parameter values

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 cell in the **Value** column is displayed with a white background, you can edit its value.

> ℹ️ While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while program run has been interrupted or aborted.
>
> **Further information:** "Status overview on the TNC bar", Page 155
>
> This status is reached after an NC block has been executed, for example in **Single Block** mode
>
> The following Q and QS parameters cannot be edited in the **Q parameter list** window:
>
> - Variable range from 100 to 199, because there might be interferences with special functions in the control.
> - Variable range from 1200 to 1399, because there might be interferences with machine manufacturer-specific functions.
>
> **Further information:** "Variable types", Page 1008

The following search options are available in the **Q parameter list** window:

- Search the entire table for any strings
- Search the **NR** column for a unique variable number

Further information: "Searching the Q parameter list window", Page 1011

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.

Searching the Q parameter list window

To search the **Q parameter list** window:

▶ Select any cell with a gray background

▶ Enter the desired string

> The control opens an input field and searches the column of the selected cell for this string.

> The control marks the first result that starts with the search string.

▼ ▶ Select the next result, if necessary

> ℹ The control displays an input field above the table. Alternatively, you can use this input field to navigate to a unique variable number. To select the input field, press the **GOTO** 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. Using variables outside the recommended ranges can lead to intersections and thus, undesired behavior. Danger of collision during machining!

▶ Only use variable ranges recommended by HEIDENHAIN

▶ Do not use pre-assigned variables

▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers

▶ Check the machining sequence using the simulation

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. If the previous value is kept, there is a danger of collision!

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

Further information: "Preassigned Q parameters", Page 1013

■ 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 1160

■ Variables 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. The control can calculate numerical values up to 10^{10}.

■ Using the **SET UNDEFINED** syntax element, you can assign the **undefined** status to your variables.

For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

If you use an undefined Q parameter in the calculation steps of your NC program, the control will display an error message and stop the program run.

Further information: "Assigning the Undefined status to a variable", Page 1022

■ The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, some decimal numbers cannot be represented with a binary value that is 100% exact (rounding error).

If you use calculated variable values for jump commands or positioning moves, you must keep this in mind.

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 saves the QR parameters in the following path: **SYS:\runtime\sys.cfg**. The **SYS:** 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 1829

24.2.2 Preassigned Q parameters

For example, the control assigns the following values to the Q parameters **Q100** to **Q199**:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Measurement results from touch-probe cycles

The control saves the values of the 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 the Q parameters **Q100** to **Q107**.

Active tool radius: Q108

The control assigns the value of the active tool radius to the Q parameter **Q108**.

The active tool radius is calculated from the following values:

- Tool radius **R** from the tool table
- Delta value **DR** from the tool table
- Delta value **DR** from the NC program, if a compensation table or tool call is used

> **i** The control will remember the active tool radius even after a restart of the control.

Further information: "Tool data", Page 256

Tool axis: Q109

The value of the Q parameter **Q109** depends on the current tool axis:

Q parameters	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

Further information: "Designation of the axes of milling machines", Page 194

Spindle status: Q110

The value of the Q parameter **Q110** depends on the M function last activated for the spindle:

Q parameters	M function
Q110 = −1	No spindle status defined
Q110 = 0	**M3** Switch spindle on clockwise
Q110 = 1	**M4** Switch spindle on counterclockwise
Q110 = 2	**M5** after **M3** Stop the spindle
Q110 = 3	**M5** after **M4** Stop the spindle

Further information: "Miscellaneous Functions", Page 963

Coolant on/off: Q111

The value of the Q parameter **Q111** depends on the M function for the coolant on/off function that was last activated:

Q parameters	M function
Q111 = 1	**M8** Switch coolant supply on
Q111 = 0	**M9** Switch coolant supply off

Overlap factor: Q112

The control assigns the overlap factor for pocket milling to the Q parameter **Q112**.

Further information: "Milling Cycles", Page 535

Unit of measure in the NC program Q113

The value of the Q parameter **Q113** depends on the unit of measure selected in the NC program. In case of program nesting (e.g., with **CALL PGM**), the control will use the unit of measure defined for the main program:

Q parameters	Unit of measure of the main program
Q113 = 0	Metric system (mm)
Q113 = 1	Imperial system (inch)

Tool length: Q114

The control assigns the value of the active tool length to the Q parameter **Q114**.

The active tool length is calculated from the following values:

- Tool length **L** from the tool table
- Delta value **DL** from the tool table
- Delta value **DL** from the NC program, if a compensation table or tool call is used

ℹ The control remembers the active tool length even after a restart of the control.

Further information: "Tool data", Page 256

Calculated coordinates of the rotary axes: Q120 to Q122

The control assigns the calculated coordinates of the rotary axes to the Q parameters **Q120** to **Q122**:

Q parameters	Rotary axis coordinates
Q120	AXIS ANGLE IN THE A AXIS
Q121	AXIS ANGLE IN THE B AXIS
Q122	AXIS ANGLE IN THE C AXIS

Measurement results from touch-probe cycles

The control assigns the measurement result of a programmable touch-probe cycle to the following Q parameters.

ℹ The help graphics of the touch-probe cycles show whether the control saves a measurement result in a variable or not.
Further information: "The Help workspace", Page 1158

Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291

Q parameters Q115 and Q116 for automatic tool measurement

The control assigns the deviation of the actual value from the nominal value in automatic tool measurements (e.g., with a TT 160) to the Q parameters **Q115** and **Q116**:

Q parameters	Deviation of actual from nominal value
Q115	Tool length
Q116	Tool radius

> **ℹ** After probing, the Q parameters **Q115** and **Q116** might contain other values.

Q parameters Q115 to Q119

The control assigns the coordinate axis values after probing to the Q parameters **Q115** to **Q119**:

Q parameters	Axis coordinates
Q115	**TOUCH POINT IN X**
Q116	**TOUCH POINT IN Y**
Q117	**TOUCH POINT IN Z**
Q118	**TOUCH POINT 4TH AXIS** (e.g., A axis) The machine manufacturer defines the 4th axis
Q119	**TOUCH POINT 5TH AXIS** (e.g., B axis) The machine manufacturer defines the 5th axis

> **ℹ** For these Q parameters, the control does not take the radius and length of the stylus into account.

Q parameters Q141 to Q149

The control assigns the measured actual values to the Q parameters **Q141** to **Q149**:

Q parameters	Measured actual values
Q141	**MEASURED ERROR A AXIS**
Q142	**MEASURED ERROR B AXIS**
Q143	**MEASURED ERROR C AXIS**
Q144	**ERROR OF OPTIM. A AXIS**
Q145	**ERROR OF OPTIM. B AXIS**
Q146	**ERROR OF OPTIM. C AXIS**
Q147	**OFFSET IN A AXIS**
Q148	**OFFSET IN B AXIS**
Q149	**OFFSET IN C AXIS**

Q parameters Q150 to Q160

The control assigns the measured actual values to the Q parameters **Q150** to **Q160**:

Q parameters	Measured actual values
Q150	MEASURED ANGLE
Q151	ACTL. VALUE, REF AXIS
Q152	ACTL.VALUE, MINOR AXIS
Q153	ACTUAL VALUE, DIAMETER
Q154	ACT.VAL. PCKT REF AX.
Q155	ACT.VAL. PKT MINOR AX.
Q156	ACTUAL VALUE OF LENGTH
Q157	ACTL.VAL., CENTERLINE
Q158	PROJECTD. ANGLE A AXIS
Q159	PROJECTD. ANGLE B AXIS
Q160	COORD., MEASURING AXIS Coordinate of the axis selected in the cycle

Q parameters Q161 to Q167

The control assigns the calculated deviation values to the Q parameters **Q161** to **Q167**:

Q parameters	Calculated deviation
Q161	**ERROR, CENTR, REF AX.** Deviation of center in main axis
Q162	**ERROR, CENTR, MINOR AX** Deviation of center in the secondary axis
Q163	**ERROR OF DIAMETER**
Q164	**ERROR, PCKT., REF AX.** Deviation of pocket length in the main axis
Q165	**ERROR, CENTR, MINOR AX** Deviation of pocket width in the secondary axis
Q166	**ERROR OF LENGTH** Deviation of the measured length
Q167	**ERROR OF CENTERLINE** Deviation of the centerline position

Q parameters Q170 to Q172

The control assigns the determined spatial angle values to the Q parameters **Q170** to **Q172**:

Q parameters	Determined spatial angles
Q170	SPATIAL ANGLE A
Q171	SPATIAL ANGLE B
Q172	SPATIAL ANGLE C

Q parameters Q180 to Q182

The control assigns the determined workpiece status to the Q parameters **Q180** to **Q182**:

Q parameters	Workpiece status
Q180	WORKPIECE IS GOOD
Q181	WORKPIECE NEEDS REWORK
Q182	WORKPIECE IS SCRAP

Q parameters Q190 to Q192

The control reserves the Q parameters **Q190** to **Q192** for the results of tool measurements with a laser measuring system.

Q parameters Q195 to Q198

The control reserves the Q parameters **Q195** to **Q198** for internal use:

Q parameters	Reserved for internal use
Q195	MARKER FOR CYCLES
Q196	MARKER FOR CYCLES
Q197	MARKER FOR CYCLES Cycles with position pattern
Q198	NO., LAST TCH-PRB CYC Number of the last active touch-probe cycle

Q parameter Q199

The value of the Q parameter **Q199** depends on the status of tool measurement with a tool touch probe:

Q parameters	Status of tool measurement with a tool touch probe
Q199 = 0.0	: Tool is within tolerance.
Q199 = 1.0	Tool is worn (**LTOL**/**RTOL** is exceeded)
Q199 = 2.0	Tool is broken (**LBREAK**/**RBREAK** is exceeded)

Q parameters Q950 to Q967

The control assigns the measured actual values resulting from the **14xx** touch-probe cycles to the Q parameters **Q950** to **Q967**:

Q parameters	Measured actual values
Q950	P1 measured main axis
Q951	P1 measured minor axis
Q952	P1 measured tool axis
Q953	P2 measured main axis
Q954	P2 measured minor axis
Q955	P2 measured tool axis
Q956	P3 measured main axis
Q957	P3 measured minor axis
Q958	P3 measured tool axis
Q961	Measured SPA Spatial angle **SPA** in the working plane coordinate system **WPL-CS**
Q962	Measured SPB Spatial angle **SPB** in the **WPL-CS**
Q963	Measured SPC Spatial angle **SPC** in the **WPL-CS**
Q964	Meas. basic rotation Rotational angle in the input coordinate system **I-CS**
Q965	Meas. table rotation
Q966	Measured diameter 1
Q967	Measured diameter 2

Q parameters Q980 to Q997

The control assigns the deviations calculated in connection with the **14xx** touch-probe cycles to the Q parameters **Q980** to **Q997**:

Q parameters	Measured deviations
Q980	**P1 error main axis**
Q981	**P1 error minor axis**
Q982	**P1 error tool axis**
Q983	**P2 error main axis**
Q984	**P2 error minor axis**
Q985	**P2 error tool axis**
Q986	**P3 error main axis**
Q987	**P3 error minor axis**
Q988	**P3 error tool axis**
Q994	**Error: basic rotation** Angle in the input coordinate system **I-CS**
Q995	**Meas. table rotation**
Q996	**Error: diameter 1**
Q997	**Error: diameter 2**

Q parameter Q183

The value of the Q parameter **Q183** depends on the workpiece status as measured by the 14xx touch-probe cycles:

Q parameters	Workpiece status
Q183 = −1	Not defined
Q183 = 0	Pass
Q183 = 1	Rework
Q183 = 2	Scrap

24.2.3 The 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 numerical 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 preassigned variables (e.g., the active tool radius **Q108**). Using the functions **FN 1** to **FN 5**, you can make calculations with the variable values in your NC program.

Related topics

- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1013

- Calculations using formulas

  Further information: "Formulas in the NC program", Page 1043

Description of function

The **Basic arithmetic** folder contains the following functions:

Icon	Function
=	**FN 0**: Assignment Example: **FN 0: Q5 = +60** Q5 = 60 Assign a value or the **Undefined** status
+	**FN 1**: Addition Example: **FN 1: Q1 = -Q2 + -5** Q1 = −Q2+(−5) Calculate and assign the sum of two values
−	**FN 2**: Subtraction Example: **FN 2: Q1 = +10 - +5** Q1 = +10−(+5) Calculate and assign the difference of two values.
×	**FN 3**: Multiplication Example: **FN 3: Q2 = +3 * +3** Q2 = 3*3 Calculate and assign the product of two values.
/	**FN 4**: Division Example: **FN 4: Q4 = +8 DIV +Q2** Q4 = 8/Q2 Calculate and assign the quotient of two values Restriction: You cannot divide by 0
√	**FN 5**: Square root Example: **FN 5: Q20 = SQRT 4** Q20 = √4 Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal 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, for example, you can program the characteristic workpiece dimensions as variables. When machining the individual workpieces, assign a numerical value to each variable.

11 LBL "Z1"	
12 FN 0: Q50 = +30	; Assign the value **30** to the cylinder radius **Q50**
13 FN 0: Q51 = +10	; Assign the value **10** to the cylinder height **Q51**
* - ...	
21 L X +Q50	; Result corresponds to **L X +30**

Example: Cylinder with Q parameters
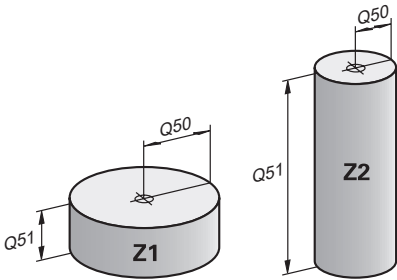
Cylinder radius:	R = Q50
Cylinder height:	H = Q51
Cylinder Z1:	Q50 = +30
	Q51 = +10
Cylinder Z2:	Q50 = +10
	Q51 = +50

Assigning the Undefined status to a variable

To assign the **Undefined** status to a variable:

▶ 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 assigns the **Undefined** status to the variable.

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**).

24.2.4 The 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 Example: **FN 6: Q20 = SIN -Q5** Q20 = sin(−Q5) Calculate and assign the sine of an angle in degrees
COS	**FN 7**: Cosine Example: **FN 7: Q21 = COS -Q5** Q21 = cos(−Q5) Calculate and assign the cosine of an angle in degrees
LEN	**FN 8**: Root of the sum of squares Example: **FN 8: Q10 = +5 LEN +4** $Q10 = \sqrt{(5^2+4^2)}$ Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
ANG	**FN 13**: angle Example: **FN 13: Q20 = +25 ANG -Q1** Q20 = arctan(25/−Q1) Calculate and assign the angle from the opposite side and the adjacent side using arctan or from the sine and cosine of the angle (0 < angle < 360°)

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, you can use fixed or variable values. The variables and numerical values in the equations can be entered with an algebraic sign.

Definition
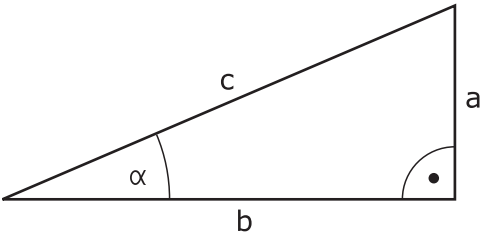
Side or trigono-metric function	Meaning
a	Opposite side The side opposite to angle **α**
b	Adjacent side The side adjacent to angle **α**
c	Hypotenuse The longest side of the triangle, opposite to the right angle
Sine	sin **α** = opposite side/hypotenuse sin **α** = a/c
Cosine	cos **α** = adjacent side/hypotenuse cos **α** = b/c
Tangent	tan **α** = opposite side/adjacent side tan **α** = a/b or tan **α** = sin **α**/cos **α**
Arc tangent	**α** = arctan(a/b) or **α** = 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 $a^2 = a \cdot a$)

$c = \sqrt{(a^2+b^2)}$

11 Q50 = ATAN (+25 / +50)	Calculate angle **α**
12 FN 8: Q51 = +25 LEN +50	Calculate side length c

24.2.5 The 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 a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

Description of function

The **Circle calculation** folder contains the following functions:

Icon	Function
⬡	**FN 23**: Circle data from three points on the circle Example: **FN 23: Q20 = CDATA Q30** The control saves the determined values in the Q parameters **Q20** to **Q22**.
⬡	**FN 24**: Circle data from four points on the circle Example: **FN 24: Q20 = CDATA Q30** The control saves the determined values in the Q parameters **Q20** to **Q22**.

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, define the variable starting from which the control is to determine the circle data from the next variables.

The coordinates of the circle data are stored in successive variables. These coordinates must be in the working plane. 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 of milling machines", Page 194

Application example

11 FN 23: Q20 = CDATA Q30	; Circle calculation with three points on the circle

The control checks the values in the Q parameters **Q30** to **Q35** and determines the circle data.

The control saves the results in the following Q parameters:

- Circle center on the main axis in the Q parameter **Q20**

  For the tool axis **Z**, the main axis is **X**
- Circle center on the secondary axis in the Q parameter **Q21**

  For the tool axis **Z**, the secondary axis is **Y**
- Circle radius in the Q parameter **Q22**

> 🛈 NC function **FN 24** uses four pairs of coordinate values and thus eight successive Q parameters.

Note

FN 23 and **FN 24** not only assign a value to the results variable to the left of the equal sign, but also to the subsequent variables.

24.2.6 The 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 variable or fixed value with another variable or fixed value. If the condition is fulfilled, the control jumps to the label programmed for 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 360

Description of function

The **Jump commands** folder contains the following functions for if-then decisions:

Icon	Function
=	**FN 9**: jump if equal Example: **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values are equal, the control jumps to the defined label.
	FN 9: jump if undefined Example: **FN 9: IF +Q1 IS UNDEFINED GOTO LBL "UPCAN25"** If the variable is undefined, the control jumps to the defined label.
	FN 9: jump if defined Example: **FN 9: IF +Q1 IS DEFINED GOTO LBL "UPCAN25"** If the variable is defined, the control jumps to the defined label.
≠	**FN 10**: jump if not equal Example: **FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values are not equal, the control jumps to the defined label.
>	**FN 11**: jump if greater than Example: **FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value is greater than the second value, the control jumps to the defined label.
<	**FN 12**: jump if less than Example: **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value is less than the second value, the control jumps to the defined label.

You can enter fixed or variable values for if-then decisions.

Unconditional jump

Unconditional jumps are jumps whose condition is always fulfilled.

11 FN 9: IF+0 EQU+0 GOTO LBL1	; Unconditional jump with **FN 9** whose condition is always fulfilled

You can use such jumps, for example, in a called NC program in which you work with subprograms. In an NC program without **M30** or **M2**, you can prevent the control from executing subprograms without a call with **LBL CALL**. As the jump address, program a label that is located directly before the program end.

Further information: "Subprograms", Page 362

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

24.2.7 Special functions for programming with variables

Output error messages with FN 14: ERROR

Application

With the **FN 14: ERROR** function, you can output error messages under program control. The messages are pre-defined 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 1972
■ Error messages in the notification menu
  Further information: "Message menu on the information bar", Page 1193

Description of function

If, during program run or during simulation, the control executes the **FN 14: ERROR** function, it will interrupt 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	Error message
0 ... 999	Machine-dependent dialog
1000 ... 2999	Control-dependent dialog
3000 ... 9999	Machine-dependent dialog
10 000 and higher	Control-dependent dialog

⚙ Refer to your machine manual.
The machine manufacturer assigns and defines the error numbers up to 999 and from 3000 to 9999.

Further information: "Preassigned error numbers for FN 14: ERROR", Page 1972

Input

11 FN 14: ERROR=1000	; Output error message with **FN 14**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 14 ERROR

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 14: ERROR	Start of syntax for error message output
Number	Number of the error message
	Fixed or variable number

Note

Please be aware that not all error messages might be available, depending on the control and the software version.

Outputting text formatted with **FN 16: F-PRINT**

Application

With the function **FN 16: F-PRINT**, you can output formatted fixed and variable numbers and texts (e.g., in order to save measuring logs).

You can output the values as follows:

- Save them to a file on the control
- Display them in a window on the screen
- Save them to a file on an external drive or USB device
- Print them to a connected printer

Related topics

■ Automatically generated measurement log for touch probe cycles

Further information: "Recording the results of measurement", Page 1478

■ Print to a connected printer

Further information: "Printers", Page 1812

Description of function

In order to output fixed or variable numbers and texts, the following is required:

■ Source file

The source file determines the contents and formatting.

■ NC function **FN 16: F-PRINT**

The control creates the output file using the NC function **FN 16**.

The maximum size of the output file is 20 kB.

Further information: "Format file for contents and formatting", Page 1029

The control creates the output file in the following cases:

■ End of program **END PGM**

■ Cancellation of program with the **NC STOP** key

■ **M_CLOSE** keyword in the format file

Further information: "Keywords", Page 1031

Format file for contents and formatting

Define the formatting and the contents of the output file in a format file with the extension ***.a**.

Further information: "The Text editor workspace", Page 839

Formatting

The formatting of the source file can be defined with the following formatting characters:

ℹ️ Please note that the input is case-sensitive.

Formatting characters	Meaning
"…"	Identifies the formatting of the contents to be output
	ℹ️ For text output, you can use the UTF-8 character set.
%F, **%D** or **%I**	Initiate the formatted output of Q, QL and QR parameters ■ **F**: Float (32-bit floating-point number) ■ **D**: Double (64-bit floating-point number) ■ **I**: Integer (32-bit integer)
9.3	Define the number of digits for the output of numerical values ■ 9: Total number of digits, including decimal separator ■ 3: Number of decimal places
%S or **%RS**	Initiate the formatted or unformatted output of a QS parameter ■ **S**: String ■ **RS**: Raw String The control takes over the following text without any changes and formatting.
,	Separate the input within a format-file line (e.g., data type and variable)
;	End of the format-file line
*	Initiate a comment line within the format file Comments are not included in the output file
%"	Output quotation marks in the output file
%%	Output a percentage sign in the output file
****	Output a backslash in the output file
\n	Output a line break in the output file
+	Output the variable value right-aligned in the output file
-	Output the variable value left-aligned in the output file

Keywords

You can define the contents of the output file with the following keywords:

Keyword	Meaning
CALL_PATH	Output the path name of the NC program that contains the **FN 16** function (e.g., **"TouchProbe: %S",CALL_PATH;**)
M_CLOSE	Close the file written to with **FN 16**
M_APPEND	Upon renewed output, append the contents of the output file to the existing output file
M_APPEND_MAX	Upon renewed output, append the contents of the output file to the existing output file until the maximum file size of 20 kB is reached (e.g., **M_APPEND_MAX20;**)
M_TRUNCATE	Upon renewed output, overwrite the output file
M_EMPTY_HIDE	Do not output blank lines for undefined or empty QS parameters in the output file
M_EMPTY_SHOW	Output blank lines for undefined or empty QS parameters and reset **M_EMPTY_HIDE**
L_ENGLISH	Outputs text only for English conversational language
L_GERMAN	Outputs 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_RUSSIAN	Outputs text only for Russian 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_KOREAN	Outputs text only for Korean conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language
L_ROMANIAN	Outputs text only for Romanian conversational language

Keyword	Meaning
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	Output the hours of the current time
MIN	Output the minutes of the current time
SEC	Output the seconds of the current time
DAY	Output the day of the current date
MONTH	Output the month of the current date
STR_MONTH	Output the month of the current date in short form
YEAR2	Output the year of the current date in two-digit format
YEAR4	Output the year of the current date in four-digit format

Input

11 FN 16: F-PRINT TNC:\mask.a / TNC:\Prot1.txt	; Output file **Prot1.txt** with the source from **Mask.a**

To navigate to this function:

Insert NC function ▶ FN ▶ Special functions ▶ FN 16 F-PRINT

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 16: F-PRINT	Start of syntax for formatted output of contents
File	Path of the source file for the output format Fixed or variable path Selection by means of a selection window
/	Separator between the two paths
File	Path under which the control saves the output file Fixed or variable path Selection by means of a selection window The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).

If you want to define variable paths, use the following syntax to enter the QS parameters:

Syntax element	Meaning
:'QS1'	Enter QS parameters with a preceding colon and between single quotation marks
:'QL3'.txt	Specify the file name extension of the target file, if required

Output options

Screen output

You can use the **FN 16** function to display messages in a window on the control screen. This allows you to display explanatory texts in such a way that the user cannot continue without reacting to them. The contents of the output text and the position in the NC program can be chosen freely. You can also output variable values.

In order to display the message on the control screen, enter **SCREEN:** as the output path.

The message is also displayed on the **FN 16** tab of the **Status** workspace.

Further information: "The FN 16 tab", Page 160

Example

11 FN 16: F-PRINT TNC:\MASKE -\MASKE1.A / SCREEN:	; Display the output file with **FN 16** on the control screen

> **i** If you want to replace the content of the window for multiple screen outputs in the NC program, define the **M_CLOSE** or **M_TRUNCATE** keyword.

The control opens the **FN16-PRINT** window for screen output. The window remains open until you close it. While the window is open, you can operate the control in the background and change to another operating mode.

You can close the window in the following ways:

- Defining the **SCLR:** output path (Screen Clear)
- Select the **OK** button
- Select the **Reset program** button
- Select a new NC program

Saving the output file

With the **FN 16** function, you can save the output files to a drive or a USB device.

To save the output file, define the path including the drive in the **FN 16** function.

Example

11 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT	; Save output file with **FN 16**

If you program the same output multiple times in the NC program, the control appends the current output to the end of the contents already output within the target file.

Printing the output file

You can use the **FN 16** function to print output files to a connected printer.

Further information: "Printers", Page 1812

The control will only print the output file if the source file ends with the **M_CLOSE** keyword.

To use the default printer, enter **Printer:** as the target path and a file name.

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 saves the file using the defined file name and the defined path. The control will not print the file name.

The control saves the file temporarily until printing is complete.

Example

11 FN 16: F-PRINT TNC:\MASKE-\MASKE1.A / PRINTER:\PRINT1	; Print output file with **FN 16**

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 define a path both in the machine parameters and in the **FN 16** function, the path in the **FN 16** function has priority.

■ If you only define the file name as the target path of the output file in the FN function, the control saves the output file in the folder of the NC program.

■ 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 specify the **%RS** function in the source file, the control takes over the defined content without formatting. This allows you to output a path specification with QS parameters, for example.

■ In the settings of the **Program** workspace, you can specify whether the control displays a screen output in a window.

   If you deactivate the screen output, the control will not display a window. The control will display the contents anyway on the **FN 16** tab of the **Status** workspace.

   Further information: "Settings in the Program workspace", Page 206

   Further information: "The FN 16 tab", Page 160

Example

Example of a format file that generates an output file with variable contents:

"TOUCHPROBE";

"%S",QS1;

M_EMPTY_HIDE;

"%S",QS2;

"%S",QS3;

M_EMPTY_SHOW;

"%S",QS4;

"DATE: %02d.%02d.%04d",DAY,MONTH,YEAR4;

"TIME: %02d:%02d",HOUR,MIN;

M_CLOSE;

Example of an NC program that defines only **QS3**:

11 Q1 = 100	; Assign the value **100** to **Q1**
12 QS3 = "Pos 1: " || TOCHAR(DAT +Q1)	; Convert the numerical value of **Q1** to an alphanumeric value and assign it to the defined string
13 FN 16: F-PRINT TNC:\fn16.a / SCREEN:	; Display the output file with **FN 16** on the control screen

Example of a screen output with two empty lines resulting from **QS1** and **QS4**:

The **FN16-PRINT** window

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 1978

- Read system data using QS parameters

  Further information: "Read system data with SYSSTR", Page 1051

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**

To navigate to this function:

Insert NC function ▶ **FN** ▶ **Special functions** ▶ **FN 18 SYSREAD**

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
	Fixed or variable number or name
ID	Group number of the system datum
	Fixed or variable number or name
NR	System data number
	Fixed or variable number or name
	Optional syntax element
IDX	Index
	Fixed or variable number or name
	Optional syntax element
.	Sub-index for system data for tools
	Fixed or variable number or name
	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 1680

Sending information from the NC program with FN 38: SEND

Application

The function **FN 38: SEND** enables you to retrieve fixed or variable 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

To navigate to this function:

Insert NC function ▶ FN ▶ Special functions ▶ FN 38 SEND

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 38: SEND	Send syntax initiator for information
Name or **QS**	Format of the text to be transmitted
	Fixed or variable name
	Output text with up to seven placeholders for the values of the variables (e.g., **%F**)
	Further information: "Format file for contents and formatting", Page 1029
/	Contents of the up to seven placeholders in the output text
	Fixed or variable number
	Optional syntax element

Notes

- Both fixed and variable numbers and texts are case-sensitive, so enter them correctly.
- To obtain **%** in the output text, enter **%%** at the desired position.

Example

In this example, you will send information to StateMonitor.

With the function **FN 38**, you can, for example, enter job data.

The following requirements must be met in order to use this function:

- StateMonitor version 1.2

  Job management with JobTerminal (option 4) is possible with StateMonitor version 1.2 or higher

- The job has been entered in StateMonitor
- Machine tool has been assigned

The following stipulations apply to this example:

- 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 required 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

You can also report the quantity of workpieces of the job.

With the **OK**, **S**, and **R** placeholders, you can specify whether the quantity of reported workpieces has been machined correctly or not.

With **A** and **I** you define how StateMonitor interprets the response. If you transfer absolute values, StateMonitor overwrites the previously valid values. If you transfer 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

24.2.8 NC functions for freely definable tables

Opening a freely definable table with FN 26: TABOPEN

Application

With the **FN 26: TABOPEN** NC function, you open a freely definable table to be written to with **FN 27: TABWRITE** or to be read from with **FN 28: TABREAD**.

Related topics

- Content and creation of freely definable tables

  Further information: "Freely definable tables *.tab", Page 1707
- Access to table values in case of low computing power

  Further information: "Table access with SQL statements", Page 1066

Description of function

Select the freely definable table to be opened by entering its path. Enter the file name with the ***.tab** extension.

Input

11 FN 26: TABOPEN TNC:\table \TAB1.TAB	; Open table with **FN 26**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 26 TABOPEN

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 26: TABOPEN	Start of syntax for opening a table
File	Path of the table to be opened
	Fixed or variable name
	Selection by means of a selection window

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** NC function, you write to the table that you previously opened with **FN 26: TABOPEN**.

Related topics

- Contents and creation of freely definable tables

  Further information: "Freely definable tables *.tab", Page 1707
- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1039

Description of function

Use the **FN 27** NC function to define the table columns to be written to by the control. Within an NC block, you can specify multiple table columns, but only one table row. You can previously define the contents to be written to the columns in variables; or you define it directly in the NC function **FN 27**.

Input

11 FN 27: TABWRITE 2/"Length,Radius" = Q2	; Write to table with **FN 27**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 27 TABWRITE

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 27: TABWRITE	Start of syntax for writing to a table
Number	Row number of the table to be written to Fixed or variable number
Name or **QS**	Column names in the table to be written to Fixed or variable name Use commas to separate multiple column names.
= or **SET UNDEFINED**	Write the table value or assign the status **undefined** **Further information:** "Preset table *.pr", Page 1710
Number, **Name**, or **QS**	Table value Fixed or variable number or name Only if **=** has been selected

Notes

- If you write to multiple columns within one NC block, you need to define the values to be written to the columns in consecutive variables.
- If you try to write to a locked or a non-existing table cell, the control displays an error message.
- If you write values to multiple columns, the control can either write only numbers or only names.
- If you define a fixed value in the **FN 27** NC function, the control will write the same value to each defined column.
- Using the **SET UNDEFINED** syntax element, you can assign the **undefined** status to your variables.

  For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

  If you use an undefined Q parameter in the calculation steps of your NC program, the control will display an error message and stop the program run.

  Further information: "Assigning the Undefined status to a variable", Page 1022

Example

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

The control writes to the columns **Radius**, **Depth**, and **D** of row **5** of the currently open table. The control writes the values from the Q parameters **Q5**, **Q6**, and **Q7** to the table.

Reading a freely definable table with FN 28: TABREAD

Application

With the **FN 28: TABREAD** NC function, you can read data from the table previously opened with **FN 26: TABOPEN**.

Related topics

■ Content and creation of freely definable tables

Further information: "Freely definable tables *.tab", Page 1707

■ Opening a freely definable table

Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1039

■ Writing a freely definable table

Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1039

Description of function

Use the **FN 28** NC function to define the table columns that the control is to read from. Within an NC block, you can specify multiple table columns, but only one table row.

Input

11 FN 28: TABREAD Q1 = 2 / "Length"	; Read table with **FN 28**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ FN 28 TABREAD

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 28: TABREAD	Start of syntax for reading from a table
Q, QL, QR, or **QS**	Variable for the source text
	The control uses this variable to save the contents from the table cells to be read.
Number	Row number in the table to be read
	Fixed or variable number
Name or **QS**	Column name in the table to be read
	Fixed or variable name
	Use commas to separate multiple column names.

Note

If you specify multiple columns in an NC block, the control saves the read values in consecutive variables of the same type (e.g., **QL1**, **QL2**, and **QL3**).

Example

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

The control reads the values of columns **X**, **Y**, and **D** from row **6** of the currently open table. The control saves the values to the Q parameters **Q10**, **Q11**, and **Q12**.

The content from the **DOC** column of the same row is saved to the **QS1** QS parameter.

24.2.9 Formulas in the NC program

Application

With the **Formula Q/QL/QR** NC function, you can define multiple arithmetic operations in a single NC block using fixed or variable values. You can also assign a single value to a variable.

Related topics

- String formula for strings

  Further information: "String functions", Page 1048

- Define a single calculation in an NC block

  Further information: "The Basic arithmetic folder", Page 1020

Description of function

As the first entry, you define the variable to which you assign the result.

To the right of the equal sign, define the arithmetic operations or a value that the control assigns to the variable.

The control provides the following options to enter formulas:

- Auto-complete

  Further information: "Entering a formula using the auto-complete function", Page 1047

- Pop-up keyboard for formula input from the action bar or from within the form

- Formula input mode of the virtual keyboard

  Further information: "Virtual keyboard of the control bar", Page 1160

Rules for formulas

Evaluation order for different operators

If a formula includes arithmetic operations involving a combination of different operators, the control evaluates the operations in a certain order. A familiar example of this is the rule that multiplication/division takes precedence over addition/subtraction (higher-level operations are performed first).

Further information: "Example", Page 1047

The control evaluates the arithmetic operations in the following order:

Order	Arithmetic operation	Operator	Arithmetic operator
1	Perform operations in parentheses first	Parentheses	**( )**
2	Note the algebraic sign	Algebraic sign	**-**
3	Calculate functions	Function	**SIN**, **COS**, **LN**, etc.
4	Exponentiation	Power	**^**
5	Multiplication and division	Point	***, /**
6	Addition and subtraction	Line	**+, -**

Further information: "Arithmetic operations", Page 1045

Order in the evaluation of equivalent operators

The control evaluates arithmetic operations with equivalent operators from left to right.

Example: 2 + 3 - 2 = (2 + 3) - 2 = 3

Exception: Concatenated powers are evaluated from right to left.

Example: 2 ^ 3 ^ 2 = 2 ^ (3 ^ 2) = 2 ^ 9 = 512

Arithmetic operations

The virtual keyboard for formula input allows you to perform the following arithmetic operations:

Button	Arithmetic operation	Operator
+	**Addition** Example: **Q10 = Q1 + Q5**	Line
-	**Subtraction** Example: **Q25 = Q7 - Q108**	Line
*	**Multiplication** Example: **Q12 = 5 * Q5**	Point
/	**Division** Example: **Q25 = Q1 / Q2**	Point
(**)**	**Parenthesize** Example: **Q12 = Q1 * (Q2 + Q3)**	Expression in parentheses
SQ	**Square** (square) Example: **Q15 = SQ 5**	Function
SQRT	**Calculate square root** (square root) Example: **Q22 = SQRT 25**	Function
SIN	**Calculate sine** Example: **Q44 = SIN 45**	Function
COS	**Calculate cosine** Example: **Q45 = COS 45**	Function
TAN	**Calculate tangent** Example: **Q46 = TAN 45**	Function
ASIN	**Calculate arcsine** Inverse function of sine The control determines the angle from the ratio of the opposite side to the hypotenuse. Example: **Q10 = ASIN (Q40 / Q20)**	Function
ACOS	**Calculate arccosine** Inverse function of cosine The control determines the angle from the ratio of the adjacent side to the hypotenuse. Example: **Q11 = ACOS Q40**	Function
ATAN	**Calculate arctangent** Inverse function of tangent The control determines the angle from the ratio of the opposite side to the adjacent side. Example: **Q12 = ATAN Q50**	Function

Button	Arithmetic operation	Operator
^	**Exponentiation** Example: **Q15 = 3 ^ 3**	Power
PI	**Use the "pi" constant** π = 3.14159 Example: **Q15 = PI**	
LN	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG	**Negate** Multiply by −1 Example: **Q2 = NEG Q1**	Function
INT	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42**	Function
	> ℹ The **INT** function does not round off—it simply truncates the decimal places.	
	Input: **0…999999999**	
ABS	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN	**Check the algebraic sign** Example: **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 remainder)** Example: **Q12 = 400 % 360** Result: **Q12 = 40**	Function

Further information: "The Basic arithmetic folder", Page 1020
Further information: "The Trigonometric functions folder", Page 1022
You can also define arithmetic operations for strings.
Further information: "String functions", Page 1048

Entering a formula using the auto-complete function

To enter a formula using the auto-complete function:

> Insert
> NC function

- ▶ Select **Insert NC function**
- ＞ The control opens the **Insert NC function** window.
- ▶ Select **Formula**
- ▶ Define a variable for the result
- ▶ Confirm your input
- ▶ Select the arithmetic operation (e.g., **SIN**)
- ▶ Enter the desired value
- ▶ Press the spacebar
- ＞ The control displays the currently available arithmetic operations.
- ▶ Select the desired arithmetic operation
- ▶ Enter the desired value
- ▶ If required, press the spacebar again
- ▶ If required, select the desired arithmetic operation
- ▶ Complete the NC block once all required data has been entered

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)	; Result = 0.5

- ▪ 1st calculation: Perform operations in parentheses first: 50 − 20 = 30
- ▪ 2nd calculation: Calculate sine of 30 = 0.5

24.3 String functions

Application

The string functions allow you to define and process strings using QS parameters (e.g., in order to create variable logs with **FN 16: F-PRINT**). In computing, a string designates an alphanumerical sequence of characters.

Related topics

- Ranges of variables

  Further information: "Variable types", Page 1008

Description of function

You can assign up to 255 characters to a QS parameter.

The following characters are permitted within QS parameters:

- Characters
- Numbers
- Special characters, for example **?**
- Control characters, for example **** for paths
- Spaces

The values of QS parameters can be processed or checked with the **Formula Q/QL/QR** and **String formula QS** NC functions.

Syntax	NC function	Higher-level NC function
DECLARE STRING	Assign an alphanumeric value to a QS parameter **Further information:** "Assigning an alphanumeric value to a QS parameter", Page 1053	
STRING FORMULA	Concatenate contents of QS parameters and assign them to a QS parameter **Further information:** "Concatenation of alphanumeric values", Page 1054	**String formula QS**
TONUMB	Convert the alphanumeric value of a QS parameter to a numerical value and assign it to a Q, QL, or QR parameter **Further information:** "Converting alphanumeric values to numerical values ", Page 1054	**Formula Q/QL/QR**
TOCHAR	Convert a numerical value to an alphanumeric value and assign it to a QS parameter **Further information:** "Converting numerical values to alphanumeric values", Page 1055	**String formula QS**
SUBSTR	Copy a substring from a QS parameter and assign it to a QS parameter **Further information:** "Copying a substring from a QS parameter", Page 1055	**String formula QS**
SYSSTR	Read system data and assign the contents to a QS parameter **Further information:** "Read system data with SYSSTR", Page 1051	**String formula QS**
INSTR	Search for a substring in a QS parameter and assign the retrieved characters to a Q, QL, or QS parameter **Further information:** "Searching for a substring within QS parameter contents", Page 1055	**Formula Q/QL/QR**
STRLEN	Determine the string length of a QS parameter and assign it to a Q, QL, or QR parameter **Further information:** "Determining the number of characters in QS parameter contents", Page 1055	**Formula Q/QL/QR**
STRCOMP	Compare QS parameters in ascending lexical order and assign the result to a Q, QL, or QR parameter **Further information:** "Comparing the lexical order of two alphanumerical strings", Page 1056	**Formula Q/QL/QR**

Syntax	NC function	Higher-level NC function
CFGREAD	Read the content of a machine parameter and assign it to a QS parameter **Further information:** "Accepting the contents of a machine parameter", Page 1057	▪ **String formula QS** ▪ **Formula Q/QL/QR**

The control provides the following options to enter formulas:

■ Auto-complete

Further information: "Entering a formula using the auto-complete function", Page 1047

■ Pop-up keyboard for formula input from the action bar or from within the form

■ Formula input mode of the virtual keyboard

Further information: "Virtual keyboard of the control bar", Page 1160

Read system data with SYSSTR

With the **SYSSTR** NC function, you can read system data and save the contents in QS parameters. Select the system datum by means of a group number **(ID)** and a number **(NR)**.

Optionally, you can enter **IDX** and **DAT**.

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 currently executed NC program
	3	Path of the NC program selected with Cycle **12 PGM CALL**
	10	Path of the NC program selected with **SEL PGM**
Channel data, 10025	1	Name of the current channel (e.g., **CH_NC**)
Values programmed in the tool call, 10060	1	Current tool name
		ℹ The NC function saves the tool name only if the tool has been called using its tool name.
Current system time, 10321	1 to 16, 20	■ 1: D.MM.YYYY h:mm:ss
		■ 2: D.MM.YYYY h:mm
		■ 3: D.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: 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.

Group name, ID no.	Number	Meaning
Touch-probe data, 10350	50	Type of the active TS workpiece touch probe
	70	Type of the active TT tool touch probe
	73	Name of the active TT workpiece touch probe from the **activeTT** machine parameter
Data for pallet machining, 10510	1	Name of the pallet being machined
	2	Path of the currently selected pallet table
NC software version, 10630	10	Number of the NC software version
Tool data, 10950	1	Current tool name
	2	Content of the **DOC** column of the current tool
	3	AFC control settings of the current tool
	4	Tool-carrier kinematics of the current tool

Reading machine parameters with CFGREAD

With the **CFGREAD** NC function, you can read out machine parameter contents of the control as numerical or alphanumeric values. The read-out numerical values are always given in metric form.

To read a machine parameter, you need to determine the following contents in the configuration editor of the control:

Icon	Type	Meaning
📁	**Key**	Group name of the machine parameter The group name can be specified optionally
📁	**Entity**	Parameter object The name always begins with **Cfg**
▢	**Attribute**	Name of the machine parameter
▢	**Index**	List index of the machine parameter The list index can be specified optionally

ℹ️ You can change the display of the existing parameters in the configuration editor for the machine parameter. By default, the parameters are displayed with short, explanatory texts.

Each time you want to read out a machine parameter with the **CFGREAD** NC function, you must first define a QS parameter with attribute, entity and key.

Further information: "Accepting the contents of a machine parameter", Page 1057

24.3.1 Assigning an alphanumeric value to a QS parameter

Before you can use and process alphanumeric values, you need to assign characters to the QS parameters. Use the **DECLARE STRING** command to do so.

To assign an alphanumeric value to a QS parameter:

Insert
NC function

▶ Select **Insert NC function**
> The control opens the **Insert NC function** window.
▶ Select **DECLARE STRING**
▶ Define a QS parameter for the result
▶ Select **Name**
▶ Enter the desired value
▶ End the NC block
▶ Execute the NC block
> The control saves the entered value in the target parameter.

In this example, the control assigns an alphanumeric value to the QS parameter **QS10**.

11 DECLARE STRING QS10 = "workpiece"	; Assign alphanumeric value to **QS10**

24.3.2 Concatenation of alphanumeric values

With the **||** concatenation operator, you can concatenate the contents of multiple QS parameters. This allows you to combine fixed and variable alphanumeric values.

To concatenate the contents of multiple QS parameters:

Insert
NC function
▶ Select **Insert NC function**
> The control opens the **Insert NC function** window.
▶ Select **String formula QS**
▶ Define a QS parameter for the result
▶ Confirm your input

⌫
▶ Press the backspace key
> The control deletes the quotation marks.
▶ Select **QS**
▶ Enter the variable number

▶ Press the spacebar
> The control displays the currently available syntax elements.
▶ Select concatenation operator **||**
▶ Select **QS**
▶ Enter the variable number
▶ End NC block
> The control saves the substrings after execution consecutively as an alphanumeric value in the target parameter.

In this example, the control concatenates the contents of the QS parameters **QS12** and **QS13**. The alphanumeric value is assigned to the QS parameter **QS10**.

11 QS10 = QS12 || QS13	; Concatenate contents of **QS12** and **QS13** and assign them to the QS parameter **QS10**

Parameter contents:
- **QS12**: **Status:**
- **QS13**: **Scrap**
- **QS10**: **Status: Scrap**

24.3.3 Converting alphanumeric values to numerical values

With the **TONUMB** NC function, you save exclusively numeric characters from a QS parameter to a different variable type. Then, you can use these values in calculations.

In this example, the control converts the alphanumeric value of the QS parameter **QS11** to a numerical value. This value is assigned to the Q parameter **Q82**.

11 Q82 = TONUMB (SRC_QS11)	; Convert alphanumeric value from **QS11** to a numerical value and assign it to **Q82**

24.3.4 Converting numerical values to alphanumeric values

With the **TOCHAR** NC function, you can save the content of a variable in a QS parameter. The saved content can, for example, be concatenated with other QS parameters.

In this example, the control converts the numerical value of the Q parameter **Q50** to an alphanumeric value. The control assigns this value to the QS parameter **QS11**.

11 QS11 = TOCHAR (DAT+Q50 DECIMALS3)	; Convert a numerical value from **Q50** to an alphanumeric value and assign it to the QS parameter **QS11**

24.3.5 Copying a substring from a QS parameter

With the **SUBSTR** NC function, you can save a defined substring from a QS parameter to another QS parameter. For example, you can use this NC function to extract the file name from an absolute file path.

In this example, the control saves a substring of the QS parameter **QS10** to the QS parameter **QS13**. Using the **BEG2** syntax element, you define that the control ignores the first two characters and starts copying from the third character. With the **LEN4** syntax element, you define that the control copies the next four characters.

11 QS13 = SUBSTR (SRC_QS10 BEG2 LEN4)	; Assign substring from **QS10** to the QS parameter **QS13**

24.3.6 Searching for a substring within QS parameter contents

With the **INSTR** NC function , you can check whether a particular substring is contained within a QS parameter. This allows you to determine, for example, whether the concatenation of multiple QS parameters was successful. For the check, you must indicate two QS parameters. The control searches the first QS parameter for the content of the second QS parameter.

If the substring is found, the control saves the number of characters until it reaches the occurrence of the substring in the result parameter. If multiple occurrences are found, the result is identical because the control saves the first one.

If the substring searched for is not found, the control saves the total number of characters in the result parameter.

In this example, the control searches the QS parameter **QS10** for the string saved in **QS13**. The search starts from the third character. When counting the characters, the control starts from zero. The control assigns the occurrence to the Q parameter **Q50** as a number of characters.

37 Q50 = INSTR (SRC_QS10 SEA_QS13 BEG2)	; Search **QS10** for substring from **QS13**

24.3.7 Determining the number of characters in QS parameter contents

The **STRLEN** NC function determines the number of characters in QS parameter contents. With this NC function, you can, for example, determine the length of a file path.

If the selected QS parameter has not been defined, the control returns the value **-1**.

In this example, the control determines the number of characters in the QS parameter **QS15**. The numerical value of the number of characters is assigned to the Q parameter **Q52**.

11 Q52 = STRLEN (SRC_QS15)	; Determine the number of characters in **QS15** and assign it to **Q52**

24.3.8 Comparing the lexical order of two alphanumerical strings

With the **STRCOMP** NC function, you can compare the lexical order of the content of two QS parameters.

The control returns the following results:

- **0**: The content of the two parameters is identical
- **-1**: In the lexical order, the content of the first QS parameter comes **before** the content of the second QS parameter
- **+1**: In the lexical order, the content of the first QS parameter comes **after** the content of the second QS parameter

The lexical order is as follows:

1 Special characters (e.g., ?_)
2 Numerals (e.g., 123)
3 Uppercase letters (e.g., ABC)
4 Lowercase letters (e.g., abc)

> **i** Starting from the first character, the control proceeds until the contents of the QS parameters differ from each other. If the contents differ starting from, for example, the fourth digit, the control aborts the check at this point.
>
> Shorter contents with identical strings are displayed first in the order (e.g., abc before abcd).

In this example, the control compares the lexical order of **QS12** and **QS14**. The result is assigned to the Q parameter **Q52** as a numerical value.

11 Q52 = STRCOMP (SRC_QS12 SEA_QS14)	; Compare the lexical order of the values of **QS12** and **QS14**

24.3.9　Accepting the contents of a machine parameter

Depending on the content of the machine parameter, you can use the **CFGREAD** NC function to take over alphanumeric values to QS parameters or numerical values to Q, QL or QR parameters.

In this example, the control saves the overlap factor from the **pocketOverlap** machine parameter as a numerical value in a Q parameter.

Specified settings in the machine parameters:

- **ChannelSettings**
- **CH_NC**
  - **CfgGeoCycle**
    - **pocketOverlap**

Example

11 QS11 = "CH_NC"	; Assign the key to the QS parameter **QS11**
12 QS12 = "CfgGeoCycle"	; Assign the entity to the QS parameter **QS12**
13 QS13 = "pocketOverlap"	; Assign the attribute to the QS parameter **QS13**
14 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out the contents of the machine parameter

The **CFGREAD** NC function contains the following syntax elements:

- **KEY_QS**: Group name (key) of the machine parameter

  > ℹ️　It no group name is available, define a blank value for the corresponding QS parameter.

- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Further information: "Reading machine parameters with CFGREAD", Page 1052

Note

If you use the **String Formula QS** NC function, the result is always an alphanumeric value. If you use the **Formula Q/QL/QR** NC function, the result is always a numerical value.

24.4 Defining counters with FUNCTION COUNT

Application

With the **FUNCTION COUNT** NC function, you control a counter from within the NC program. This counter allows you, for example, to define a target count up to which the control is to repeat the NC program.

Description of function

The counter reading remains the same 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 163

Input

| 11 FUNCTION COUNT TARGET5 | ; Set the target count of the counter to **5** |

Insert NC function ▶ All functions ▶ FN ▶ FUNCTION COUNT

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 1058

Counting functions

The **FUNCTION COUNT** NC function provides the following counter functions:

Syntax	Function
INC	Increase the counter by 1
RESET	Reset the counter
ADD	Increase the counter by a defined value Fixed or variable number or name Input: **0...9999**
SET	Assign a defined value to the counter Fixed or variable number or name Input: **0...9999**
TARGET	Define the target count to be reached Fixed or variable number or name Input: **0...9999**
REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet Fixed or variable number or name

Notes

<table>
<tr><td colspan="2">NOTICE</td></tr>
<tr><td colspan="2">Caution: Data may be lost!</td></tr>
<tr><td colspan="2">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.</td></tr>
<tr><td colspan="2">▶   Please check prior to machining whether a counter is active.</td></tr>
</table>

- The machine manufacturer uses the optional machine parameter **CfgNcCounter** (no. 129100) to define whether you can edit the counter.
- You can engrave the current counter reading with Cycle **225 ENGRAVING**.
  Further information: "Cycle 225 ENGRAVING ", Page 681

24.4.1 Example

11 FUNCTION COUNT RESET	; Reset counter value
12 FUNCTION COUNT TARGET10	; Define the target count of machining operations
13 LBL 11	; Set a jump label
* - …	; Execute the machining operation
21 FUNCTION COUNT INC	; Increase the counter reading by 1
22 FUNCTION COUNT REPEAT LBL 11	; Repeat the machining operation until the target count has been reached

24.5 Program defaults for cycles

24.5.1 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		Call	Further information
100	**GENERAL**	**DEF**-active	Page 1062
	Definition of generally valid cycle parameters		
	■ **Q200 SET-UP CLEARANCE**		
	■ **Q204 2ND SET-UP CLEARANCE**		
	■ **Q253 F PRE-POSITIONING**		
	■ **Q208 RETRACTION FEED RATE**		
105	**DRILLING**	**DEF**-active	Page 1063
	Definition of specific drilling cycle parameters		
	■ **Q256 DIST FOR CHIP BRKNG**		
	■ **Q210 DWELL TIME AT TOP**		
	■ **Q211 DWELL TIME AT DEPTH**		
110	**POCKET MILLING**	**DEF**-active	Page 1064
	Definition of specific pocket-milling cycle parameters		
	■ **Q370 TOOL PATH OVERLAP**		
	■ **Q351 CLIMB OR UP-CUT**		
	■ **Q366 PLUNGE**		
111	**CONTOUR MILLING**	**DEF**-active	Page 1065
	Definition of specific contour-milling cycle parameters		
	■ **Q2 TOOL PATH OVERLAP**		
	■ **Q6 SET-UP CLEARANCE**		
	■ **Q7 CLEARANCE HEIGHT**		
	■ **Q9 ROTATIONAL DIRECTION**		
125	**POSITIONING**	**DEF**-active	Page 1065
	Definition of the positioning behavior with **CYCL CALL PAT**		
	■ **Q345 SELECT POS. HEIGHT**		
120	**PROBING**	**DEF**-active	Page 1066
	Definition of specific touch probe cycle parameters		
	■ **Q320 SET-UP CLEARANCE**		
	■ **Q260 CLEARANCE HEIGHT**		
	■ **Q301 MOVE TO CLEARANCE**		

24.5.2 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

24.5.3 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. There is a danger of collision!

▶ 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**.

24.5.4 Global data valid everywhere

The parameters are valid for all **2xx** machining cycles and touch probe cycles **451**, **452**

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 the tool and the 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

24.5.5 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**

24.5.6 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

24.5.7 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). This 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

24.5.8 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

24.5.9 Global data for probing functions

The parameters apply to all touch-probe cycles **4xx** and **14xx** as well as the Cycles **271**, **1271**, **1272**, **1273**, **1274**, **1278**

Help graphic	Parameter
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **–99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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

24.6 Table access with SQL statements

24.6.1 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: "NC functions for freely definable tables", Page 1039

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**

> **i** 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 1069
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	Page 1070
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 1073
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	Page 1078
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	Page 1079
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	Page 1081
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	Page 1082
SQL INSERT	**SQL INSERT** creates a new table row	Page 1084

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

NOTICE

Danger of collision!

If you simulate an NC program that includes SQL commands, the control might overwrite table values. Overwriting table values might result in incorrect positioning of the machine. There is a danger of collision.

▶ Program NC programs in such a way that SQL commands are not executed during simulation
▶ Use **FN18: SYSREAD ID992 NR16** to check whether the NC program is active in a different operating mode or in **Simulation**

■ 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.

24.6.2 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
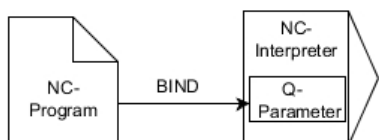
Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.

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

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ SQL ▶ SQL BIND

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
Name or **QS**	Table name and table column, separated by **.** or QS parameter with definition Fixed or variable name Optional syntax element

Notes

- Enter the path of the table or a synonym as the table name.

  Further information: "Executing SQL statements with SQL EXECUTE", Page 1073

- 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.

24.6.3 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
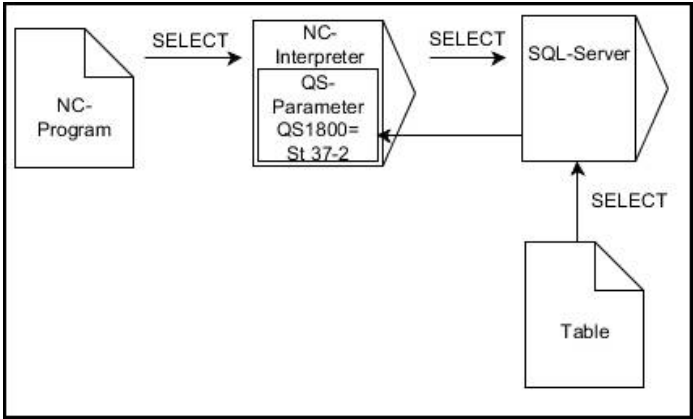
Black arrows and associated syntax show internal processes of **SQL SELECT**

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**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ SQL ▶ SQL SELECT

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL SELECT	Syntax initiator for the **SELECT** SQL command
Q, **QL**, **QR**, **QS**, or **Q REF**	Variable in which the control stores the result
Name 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) Fixed or variable name

Notes

■ You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** statement.

■ After the **WHERE** syntax element, you can define the comparison value, which can also be a variable. If you use Q, QL, or QR parameters for the comparison, the control will round the defined value to the next integer. If you use a QS parameter, the control will use the exact value you specified.

■ For the instructions within the SQL command, you can likewise use single or combined QS parameters.
Further information: "Concatenation of alphanumeric values", Page 1054

■ 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 165

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	
* - ...	

24.6.4 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
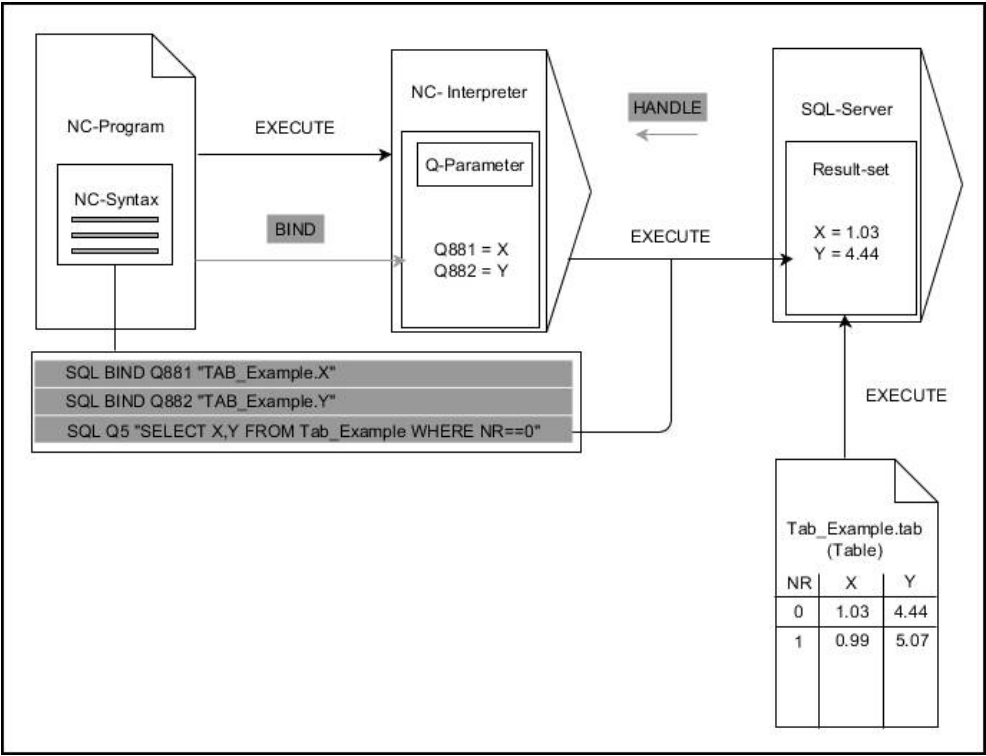
Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**. The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command.

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 table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete 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 SELECT instruction

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.

Enter the search criteria in the **SQL ... "SELECT...WHERE..."** function. 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.

Enter the ordering criteria in the **SQL ... "SELECT...ORDER BY..."** function. 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 entires

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

- If you use the **SQL EXECUTE** NC function, the control will insert the **SQL** syntax element into the NC program only.
- 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: "Concatenation of alphanumeric values", Page 1054
- After the **WHERE** syntax element, you can define the comparison value, which can also be a variable. If you use Q, QL, or QR parameters for the comparison, the control will round the defined value to the next integer. If you use a QS parameter, the control will use the exact value you specified.
- 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 165

Example

Example: selecting 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: selecting 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: selecting 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: defining 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	

24.6.5 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
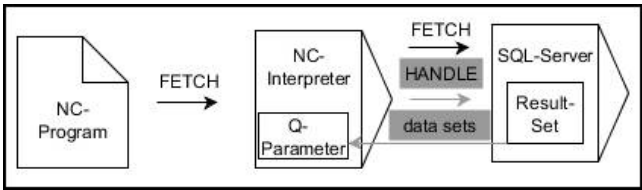
Black arrows and associated syntax indicate internal processes of **SQL FETCH**. The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command.

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

24.6.6 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
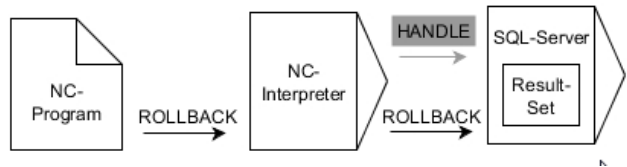
Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**. The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command.

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

24.6.7 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.
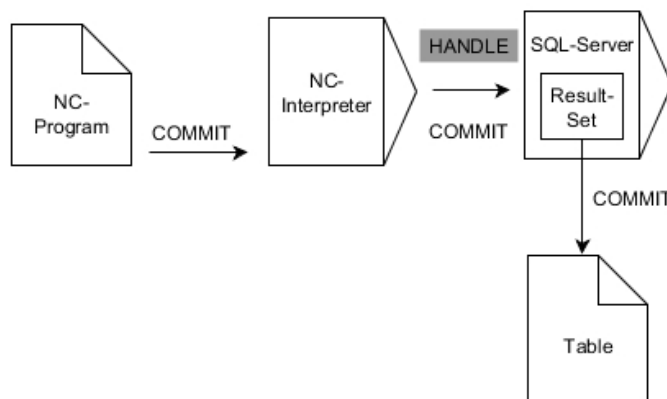
Black arrows and associated syntax indicate internal processes of **SQL COMMIT**.

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

24.6.8 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
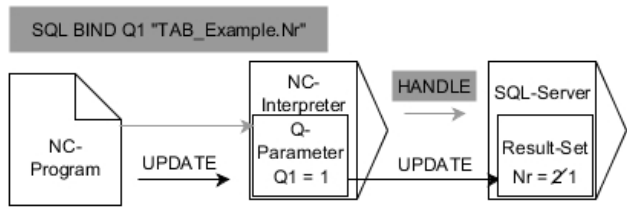
Black arrows and the associated syntax show internal **SQL UPDATE** processes. Gray arrows and the associated syntax are not directly associated with the **SQL UPDATE** command.

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

24.6.9 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
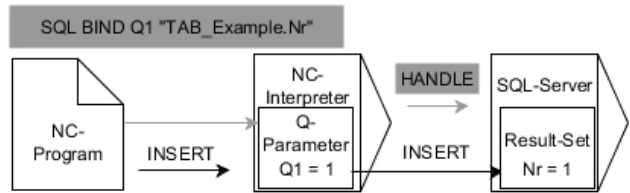
Black arrows and associated syntax indicate internal processes of **SQL INSERT**. The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command.

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 |

24.6.10 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.

> **i** 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)

> **i** 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**	

25

Graphical programming

25.1 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 independently via a separate tab or in the separate **Contour graphics** workspace. If you use graphical programming on its own tab, you cannot open any other workspaces in the **Editor** operating mode on this tab.

Description of function

The **Contour graphics** workspace is available in the **Editor** operating mode.

Screen layout
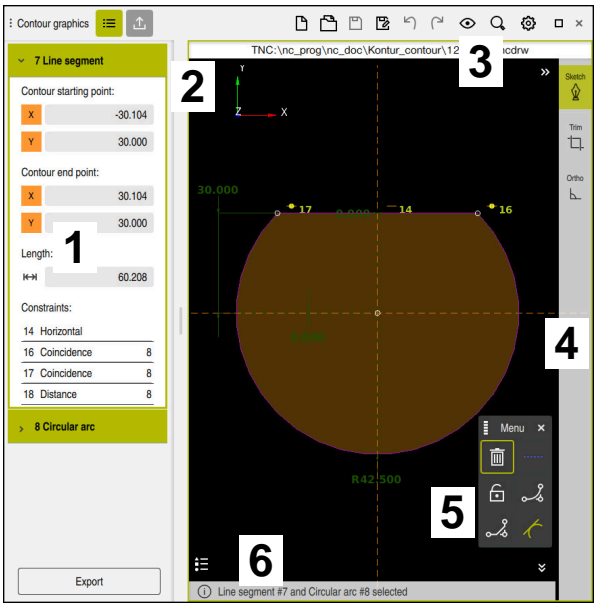
Screen layout of the **Contour graphics** workspace

The **Contour graphics** workspace contains the following areas:

1   Element information area
2   Drawing area
3   Title bar
4   Toolbar
5   Drawing functions
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 1103

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 98

Icon	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

Besides icons solely available for graphical programming, the title bar of the
Contour graphics workspace also includes general icons of the control interface.
Further information: "Icons on the control's user interface", Page 107

The control shows the following icons in the title bar:

Icon or shortcut	Meaning
↥	Open or close the **Export** column
▯ CTRL + N	**Discard** the contour
▢ CTRL + O	**Open File**
◉	Open or close the **Viewing options** selection menu
↦	**Hide dimensions**
↦	**Show dimensions**
⌀	**Hide restrictions**
⌀	**Show restrictions**
⊕	**Hide reference axes**
⊕	**Show reference axes**
⌕	Open or close the **Scaling options** selection menu
⊡	**Drawing area** Scale the view to the drawing area You can define the size of the drawing area in the contour settings. **Further information:** "The Contour settings window", Page 1096
▦	**Selected elements** Scale the view to the selected elements
▫▫	**All elements** Scale the view to all elements
⚙	Open or close the **Contour settings** window **Further information:** "The Contour settings window", Page 1096

Possible colors

The control shows the elements in the following colors:

Icon	Meaning
▬	**Element** A drawn element that is not fully dimensioned is displayed 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 graphical programming 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 as brown dashed lines.
▬	**Locked element** Locked elements cannot be edited. If you want to edit a locked element, you must unlock it first. Locked elements are shown by the control as red solid lines.
▬	**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. When you select the area of a closed contour, you can round all corners of the contour.
⌣	**Chamfer**	Inserts a chamfer. When you select the area of a closed contour, you can chamfer all corners of the contour.
⊶	**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** constraint for the selected **Line segment** element. Vertical elements are automatically vertical.
—	**Horizontal**	This function sets the **Horizontal** constraint for the selected **Line segment** element. Horizontal elements are automatically horizontal.
∟	**Perpendicular**	This function sets the **Perpendicular** constraint for two selected elements of the **Line segment** type. There is an angle of 90° between perpendicular elements.
∥	**Parallel**	This function sets the **Parallel** constraint 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 constraints such as **Horizontal**.

Icon or shortcut	Designation	Meaning
		Behavior in the case of constraints:
		■ If there is a constraint, the **Line segment** without constraint is aligned with the **Line segment** with constraint.
		■ If both lines have constraints, the function cannot be applied. The dimension is over-determined.
		■ If there are no constraints, 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). First, the control checks whether there are constraints, such as a defined length.
		Behavior in the case of constraints:
		■ If there is a constraint, the element without constraint is aligned with the element with constraint.
		■ If both elements have corresponding constraints, the function cannot be applied. The dimension is over-determined.
		■ If there are no constraints, the control calculates the average value from the given dimensions.
	Tangential	This function sets the **Tangential** constraint for two marked elements of the **Line segment** and **Circular arc** or **Circular arc** and **Circular arc** types.
		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** constraint for a marked element of the **Line segment** type 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** constraint 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.
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.

Icon or shortcut	Designation	Meaning
		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.

The Contour settings window

The **Contour settings** window contains the following areas:

- **General information**
- **Sketching**
- **Export**

The control saves the settings permanently.

Only the **Plane** setting is not saved.

The 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**
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

The 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 toggle switchto select whether roundings drawn with the **RND** function are also exported as **RND** to the NC program.
CHF output	You use a toggle switch to select whether chamfers drawn with the **CHF** function are also exported as **CHF** to the NC program.

25.1.1 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 **Contour**
> The control opens the contour in a new tab.

25.1.2 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 or 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

- Define the **Contour settings** before drawing.
  Further information: "The Contour settings window", Page 1096
- 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 constraints to the drawn elements. To avoid unnecessarily complicating the design, work only with necessary constraints.
  Further information: "Icons in the drawing area", Page 1094
- 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

25.2 Importing contours into graphical programming

Application

In 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** (#9 / #4-01-1)
- 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 the contour to the **Contour graphics** workspace:

- Circular contour **CT**
  Further information: "Circular path CT", Page 313
- NC blocks with polar coordinates
  Further information: "Polar coordinates", Page 292
- NC blocks with incremental inputs
  Further information: "Incremental entries", Page 295
- Free contour programming **FK**

25.2.1 Importing contours
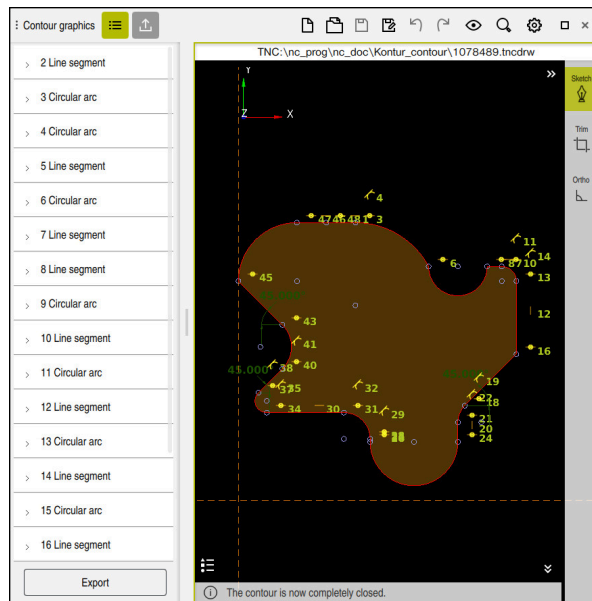
Imported contour

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 contour**
> The control opens the marked contour area in the **Contour graphics** workspace.

> ℹ You can also import contours by dragging the selected NC blocks into the open **Contour graphics** workspace. For this purpose, the control shows a green icon at the right margin of the first highlighted NC block.
>
> **Further information:** "Common gestures for the touchscreen", Page 98

Notes

- When importing a contour into graphical programming using the **Edit contour** function, all elements are initially locked. Before you begin editing the elements, you must unlock the elements.

  Further information: "Locking and unlocking elements", Page 1097

- You can edit contours graphically and export them after importing.

  Further information: "First steps in graphical programming", Page 1103

  Further information: "Exporting contours from graphical programming", Page 1100

- You can also import NC functions in conjunction with the contour for the coordinate transformation. As soon as you additionally import a transformation, the control will take it into account (e.g., mirroring with **TRANS MIRROR**).

25.3 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 1097

- First steps in graphical programming

  Further information: "First steps in graphical programming", Page 1103

Description of function

Contour starting point	
X	-34.177
Y	-25.262
Set graphically	

Contour end point	
X	-34.177
Y	-25.262
Set graphically	

Invert direction
Generate Klartext
Reset selection

Sketching

The **Export** column includes the following areas:

- **Contour starting point**

  In this area, you define the **Contour starting point**. 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**

  In this area, you define the **Contour end point**. You can set the **Contour end point** in the same way as the **Contour starting point**.

Icons or buttons

Icon or button	Meaning
Set graphically	Graphically set the Contour starting point or Contour end point
⎘	**Closed contour**
	In a closed contour, the starting and end point coincide. When you select the starting point, the control will set the end point automatically.
⎘	**Open contour**
	In an open contour, the starting and end point do not coincide.
	When you select the icon, the control closes the contour and sets the end point to the starting point automatically.
Invert direction	This function will 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: "The Contour settings window", Page 1096
	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.

25.4 First steps in graphical programming

25.4.1 Example task D1226664

START

R42.5

30

□ 100

16

5

744 650 A4

3:10

		ID number	
Text:		Change No.	C000941-05
		Phase:	Nicht-Serie

	Original drawing	Platte	Werkstoff: 3.1645
	Scale	Format	Plate	Material:
RoHS	1:1	A4		

Maße in mm / Dimensions in mm | Einzelteilzeichnung / Component Drawing | ●blanke Flächen/Blank surfaces

Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715	Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2 General tolerances ISO 2768-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015	Oberflächen nach ISO 1302 Surfaces as per ISO 1302
-0.3 +0.3			Oberflächenbehandlung: Surface treatment:

HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	Created	Responsible	Released		Version	Revision	Sheet	Page
	M-TS			D1226664-00-A-01				1 of 1
	05.09.2017			Document number				

25.4.2 Drawing a sample contour

To draw the displayed contour:

▶ Create a new contour
   Further information: "Creating a new contour", Page 1097

▶ Configure **Contour settings**

> 🛈 In the **Contour settings** window, you can define basic settings for
> drawing. For this example, you can use the default settings.

Further information: "The Contour settings window", Page 1096

▶ Draw a 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 a **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.
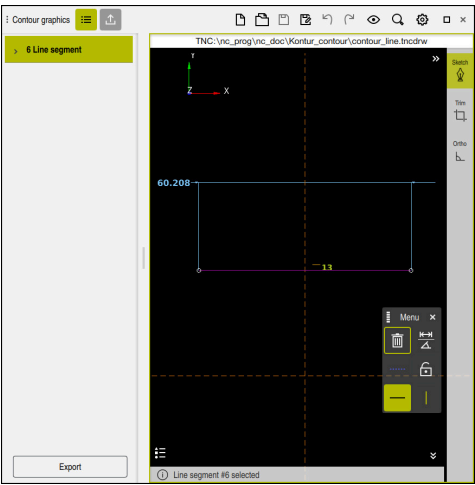
Line drawn
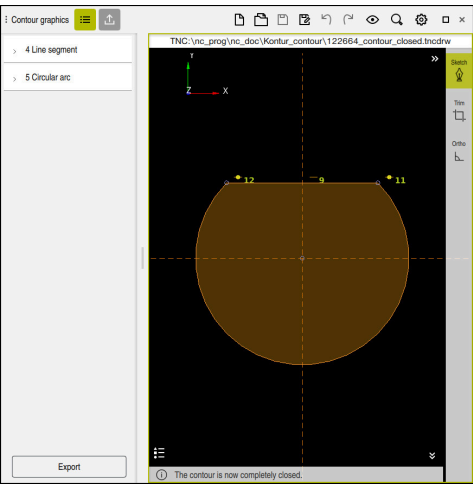
Closed contour
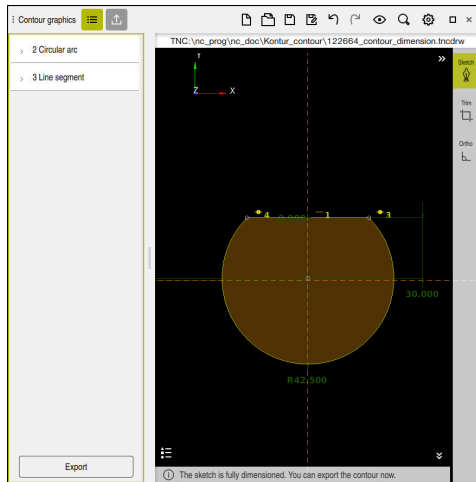
Dimensioned contour

25.4.3 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.
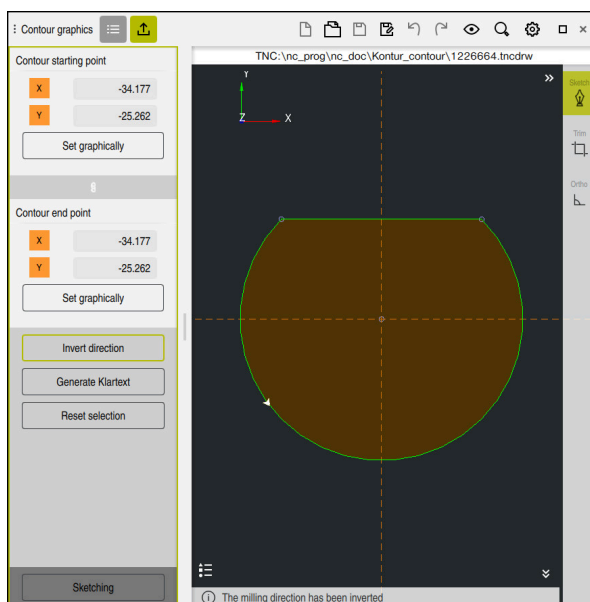
Selected contour elements in the **Export** column with defined **Milling direction**

26

Opening CAD files
with CAD Viewer

26.1 Fundamentals

Application

CAD Viewer supports the following standard file types that can be opened directly in the control:

File type	Extension	Format
STEP	*.stp and *.step	■ AP 203
		■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015
		■ ASCII
STL	*.stl	■ Binary
		■ ASCII

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 1089

Description of function

Screen layout
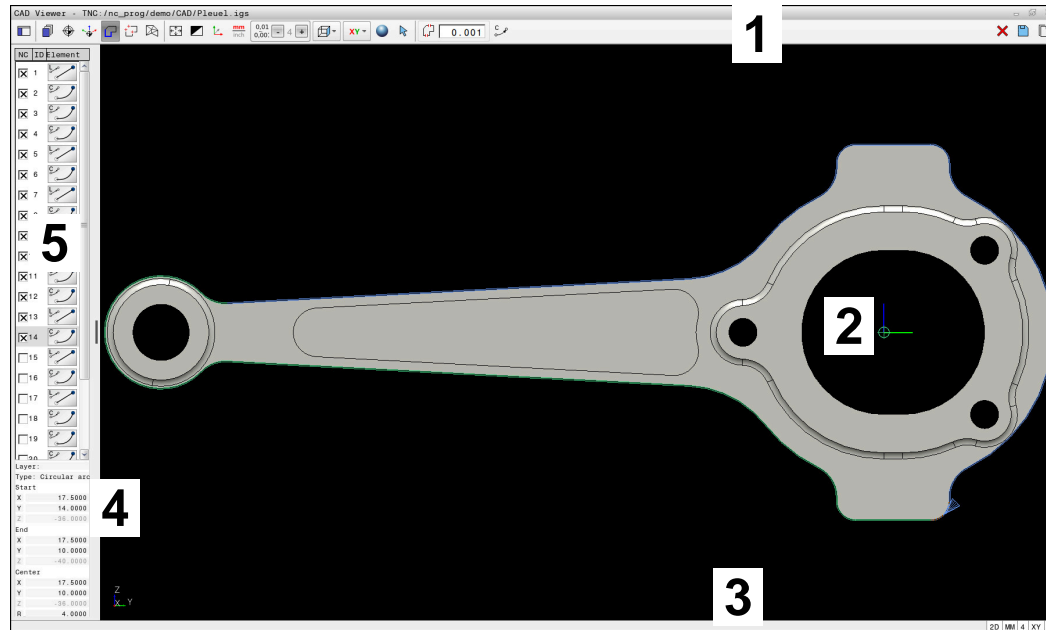
CAD file open in **CAD Viewer**

CAD Viewer consists of the following areas:

1 Menu bar
   Further information: "Menu bar icons", Page 1110

2 Graphics area
   The CAD model is displayed in the graphics window.

3 Status bar
   The status bar contains the active settings.

4 Element information area
   Further information: "Element Information area", Page 1112

5 List View area
   The List View area displays information on the active function (e.g., available layers or the position of the workpiece preset).

Menu bar icons

The menu bar contains the following icons:

Icon	Meaning
	Show sidebar
	Show, enlarge, or hide the List View area
	Display the layer
	Display the layer(s) in the List View area
	Further information: "Layer", Page 1112
	Preset
	Define the workpiece preset
	Workpiece preset has been defined
	Delete the defined workpiece preset
	Further information: "Workpiece preset in the CAD file", Page 1113
	Datum
	Set the datum
	Datum has been set
	Further information: "Workpiece datum in the CAD file", Page 1116
	Contour
	Select contour (#42 / #1-03-1)
	Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
	Positions
	Select positions (#42 / #1-03-1)
	Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
	3D mesh
	Create a 3D mesh (#152 / #1-04-1)
	Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124
	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)
	CAD Viewer performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts all values to inches.
	Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
0.01 ... 0.001	**Number of decimal places**

Icon	Meaning
	Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118 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**)
XY	**Axes** Select the working plane: ■ **XY** ■ **YZ** ■ **ZX** ■ **ZXØ** If you take over a contour or position, the control outputs the NC program in the selected working plane. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
	Toggle a 3D model between a solid model and a wire-frame model.
	"Select, add, or remove contour elements" mode ⓘ The icon shows the current mode. Clicking the icon activates the next mode. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 1118
	Undo
	Delete entire list
	Save entire list content to a file
	Copy entire list contents to clipboard The control retains the content of the clipboard only as long as **CAD Viewer** is open.

Element Information area

In the Element Information area, 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

The control always shows the **X**, **Y**, and **Z** coordinates. In 2D mode, the Z coordinate is dimmed.

Layer

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.

If the name of the layer is not shown completely in the List View area, you can use the **Show sidebar** icon to enlarge this area.

Use the **Display the layer** icon to display all the layers of the file in the List View area. Use the check box in front of the name to show and hide individual layers.

When you open a CAD file in **CAD Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- Before loading the file into the control, ensure that the name of the file contains only permitted characters.
  Further information: "Permitted characters", Page 827
- When you select a layer in the List View area, you can press the spacebar to show and hide the layer.
- **CAD Viewer** allows you to open CAD files consisting of any number of triangles.

26.2 Workpiece preset in the CAD file

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 196

Description of function

When you select the **Preset** icon, the control displays the following information in the list view area:

- Distance between the defined preset and the drawing datum
- Orientation of the working plane

The control displays values not equal to 0 in orange.
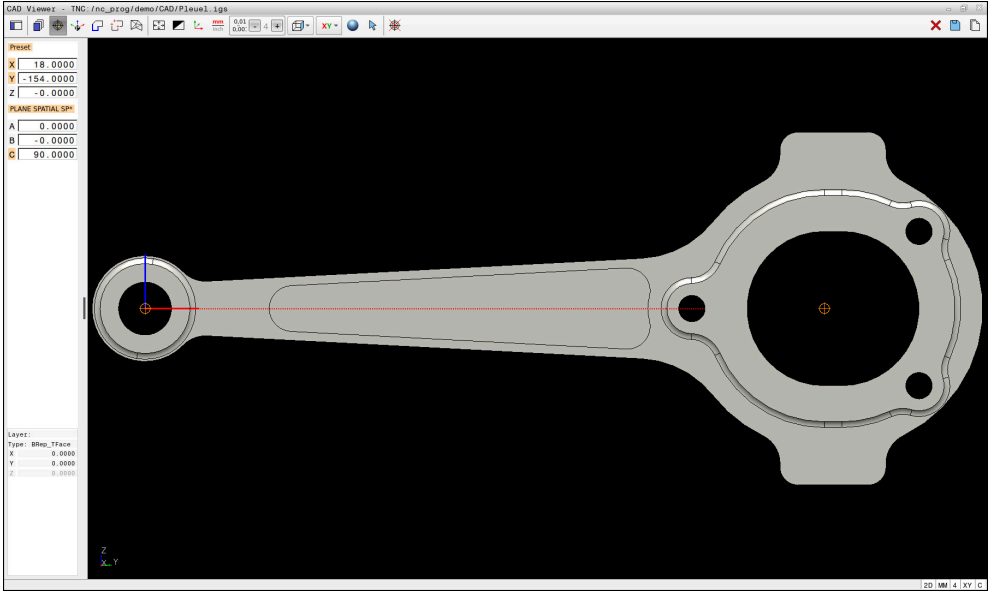
Workpiece preset in the CAD file

You can position the preset at the following locations:

- By direct numerical input in the List View area
- For lines:
    - Starting point
    - Midpoint
    - End point
- For circular arcs:
    - Starting point
    - Midpoint
    - End point
- For full circles:
    - At the quadrant transitions
    - At the center
- At the intersection between:
    - Two lines, even if the point of intersection is actually on the extension of one of the lines
    - Line and circular arc
    - 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 save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import (#42 / #1-03-1) is not available.

> **ℹ** The control retains the content of the clipboard only as long as **CAD Viewer** is open.

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.

26.2.1 Setting the workpiece preset or workpiece datum and orienting the coordinate system

> **ℹ** ■ The following instructions apply when using a mouse. You can also perform these steps with touch gestures.
> **Further information:** "Common gestures for the touchscreen", Page 98
> ■ The following instructions also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.

Setting the workpiece preset or workpiece datum on an 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 displays selectable presets for the element 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.
▶ Orient the working plane, if required

Setting the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at the intersection of 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 point of intersection of the two elements. The control marks the workpiece preset with a green symbol.
- ▶ Orient the working plane, 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.

Orienting the working plane

The following requirements must be met in order to orient the working plane:
- ▪ Preset has been defined
- ▪ There are elements next to the preset that can be used for the desired orientation

To orient the working plane:
- ▶ Select an element in the positive direction of the X axis
- ＞ The control orients the X axis.
- ＞ The control changes the **C** angle in the List View area.
- ▶ Select an element in the positive direction of the Y axis
- ＞ The control orients the Y and Z axes.
- ＞ The control changes the **A** and **C** angles in the List View area.

26.3 Workpiece datum in the CAD file

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control provides a function to define a new datum and a working plane.

Related topics
- ▪ Presets in the machine
  Further information: "Presets in the machine", Page 196

Description of function

When you select the **Datum** icon, the control displays the following information in the list view area:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the working plane

You can apply a workpiece datum set in CAD Viewer and shift it, if required by entering values directly in the List View area.

The control displays values not equal to 0 in orange.
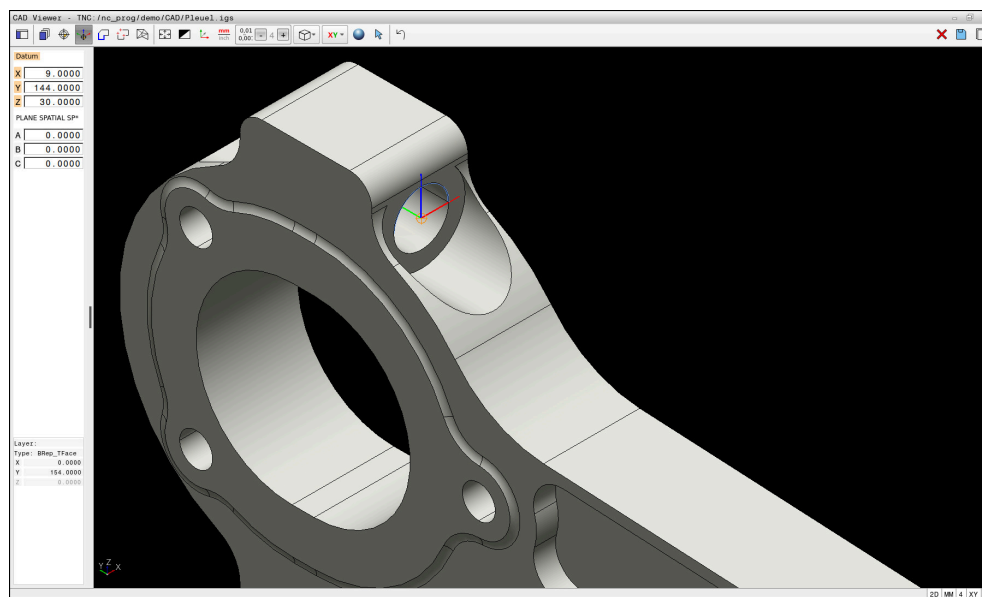
Workpiece datum for tilted machining

You can set the datum and the working plane orientation at the same locations as a preset.

Further information: "Workpiece preset in the CAD file", Page 1113

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Setting the workpiece preset or workpiece datum and orienting the coordinate system", Page 1115

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

You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import (#42 / #1-03-1) is not available.

> **i** The control retains the content of the clipboard only as long as **CAD Viewer** is open.

26.4 Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)

Application

You can open CAD files directly on the control to extract contours or machining positions from them. 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 by default contain only **L** and **CC**/**C** blocks.

Related topics

■ Using point tables
Further information: "Point tables", Page 391

Requirement

■ Software option CAD Import (#42 / #1-03-1)

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 additional software tools (e.g., **Leafpad** or **Gnumeric**).

Further information: "Opening files with additional software", Page 1881
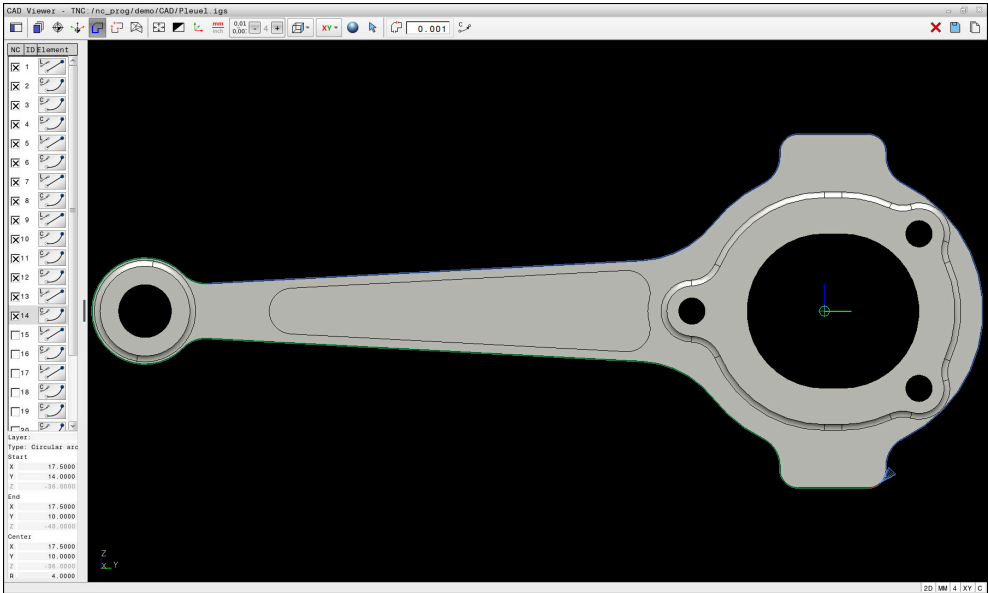
CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Meaning
	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 during drawing creation. The default setting is 0.001 mm.
C	**C or CR**
CR	You can select whether the control will output circular contours **C** or **CR** in the NC program.
	Show connections between two positions
	The control hides and displays the tool paths between the positions.
	Apply path optimization
	The control optimizes the tool traverse movement between the machining positions. When you select the icon again, the control will discard the optimization.
	Find circles according to diameter range. Load center coordinates to the position list
	The control opens the **Find circle centers by diameter range** window. You can filter the displayed data by diameter or depth values.

Applying contours

The following elements can be selected as a contour:

- Line segment
- Full circle
- Pitch circle
- Polyline
- Any curves (e.g., splines, ellipses)

Linearization

CAD Viewer linearizes all of the contours that are not in the working plane.

During linearization, **CAD Viewer** subdivides a contour into individual segments. From these segments, CAD Import creates straight lines **L** and circular arcs **C** or **CR** that are as long as possible.

Thanks to linearization, it is also possible to import contours with CAD Import that cannot be programmed with the path functions of the control, such as splines.

The higher you define the resolution by specifying decimal places, the lower is the deviation from the imported contour.

Further information: "Screen layout", Page 1109

> **ⓘ** You can prevent the linearization, for example of circles that are not in the working plane. Select the working plane in which the circle has been defined.

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: "Selecting positions", Page 1122

The following file types are available:

■ Point table (**.PNT**)
■ Klartext 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**).

> **i** **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.

Multi-selection filter settings

If you use the quick-selection function to mark positions, the **Find circle centers by diameter range** window opens. You can filter the diameter or depth values, referencing the workpiece datum, by means of the buttons below the displayed value. The control will only load the selected diameter or depth values.

The **Find circle centers by diameter range** window provides the following buttons:

Button	Meaning
|<<	■ The control shows the smallest diameter found. ■ The control shows the smallest depth found. This filter is active by default.
<<	■ The control sets the filter for the largest diameter to the value selected for the smallest diameter. ■ The control sets the filter for the largest depth to the value selected for the smallest depth.
<	■ The control shows the next smaller diameter found. ■ The control shows the next smaller depth found.
>	■ The control shows the next larger diameter found. ■ The control shows the next larger depth found.
>>	■ The control sets the filter for the smallest diameter to the value selected for the largest diameter. ■ The control sets the filter for the smallest depth to the value selected for the largest depth.
>> |	■ The control shows the largest diameter found. ■ The control shows the largest depth found. This filter is active by default.

26.4.1 Selecting and saving a contour

> **ℹ** ▪ 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 98
> ▪ Deselecting, deleting, and saving of elements works in the same way for applying 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 List View area.
> Other contour elements are shown in green.

> **ℹ** 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 List View area.
▸ Select **Save entire list content to a file**
> The control opens the **Define file name for contour program** window.
▸ Enter the desired name
▸ Select the path to the storage location
▸ Select **Save**
> The selected contour is saved as an NC program.

> **ℹ** ▪ 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

ℹ 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.

26.4.2 Selecting positions

ℹ ■ 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 98
■ Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.
Further information: "Selecting and saving a contour", Page 1121

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 area.

Multiple selection within an 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 by diameter range** window. The window shows the identified diameter and depth values.

▶ Change the filter settings as needed

▶ Select **OK**

> The control loads all positions within the selected diameter and depth ranges into the List View area.

> The control shows the traverse distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:

▶ Select **Positions**

▶ Select **Find circles according to diameter range. Load center coordinates to the position list**

> The control opens the **Find circle centers by diameter range** window. The window shows the identified diameter and depth values.

▶ Change the filter settings as needed

▶ Select **OK**

> The control loads all positions within the selected diameter and depth ranges into the List View area.

> The control shows the traverse distance between the positions.

Notes

- Set the correct unit of measure so that **CAD Viewer** shows the correct values.
- Ensure that the unit of measure used in the NC program matches that used in **CAD Viewer**. Elements that have been copied from **CAD Viewer** to the clipboard do not contain any information about the unit of measure.
- The control retains the content of the clipboard only as long as **CAD Viewer** is open.
- **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.
- 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.
- CAD Import outputs the radii of the circular arcs as comments. At the end of the generated NC blocks, CAD Import displays the smallest radius to help you select the most suitable tool.

Notes on Contour Transfer

■ If you double-click a layer in the List View area, 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 it 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:

━

26.5 Generating STL files with 3D mesh (#152 / #1-04-1)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective fixture and tool holder files, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

■ Fixture management
Further information: "Fixture management", Page 856

■ Export the simulated workpiece as an STL file
Further information: "Exporting a simulated workpiece as STL file", Page 1209

■ Using an STL file as workpiece blank
Further information: "Defining a workpiece blank with BLK FORM", Page 244

Requirement

■ Software option CAD Model Optimizer (#152 / #1-04-1)

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, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.
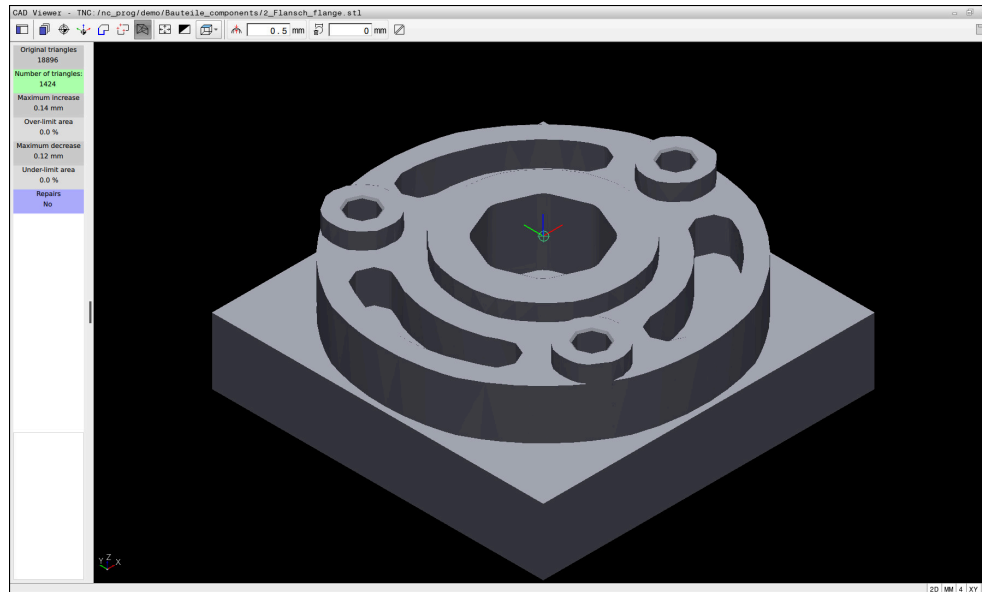
3D model in **3D mesh** mode

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 settings selected in **3D mesh** mode.

The List View area 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
	ℹ️ 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 1126
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
Repairs	Indicates whether the original model has been repaired or not

Option	Meaning
	If it has been repaired, the control indicates the type of repair (e.g., **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:

Icon	Meaning
	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
	The control shows the simplified model only.
	Original is displayed
	The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	Save
	Use this function to save the simplified 3D model with the selected settings as an STL file.

26.5.1 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 1211

⯁ ▶ Select the **Files** operating mode

▶ Select the exported STL file

> The control opens the STL file in **CAD Viewer**.

⊕ ▶ Select **Preset**

> The control displays information on the preset position in the List View area.

▶ Enter the value of the new preset in the **Preset** area (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
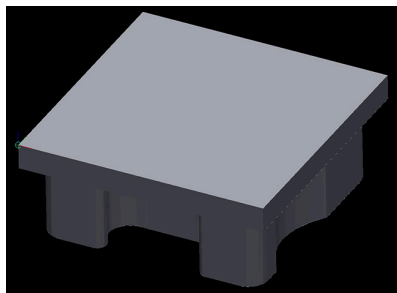
⬡ ▶ Select **3D mesh**

> 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 1126

🖫 ▶ Select **Save**

> The control opens the **Define file name for 3D mesh** window.

▶ Enter the desired name

▶ Select **Save**

> The control saves the STL file positioned for rear-face machining.

ℹ️ 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 244

27

ISO

27.1 Fundamentals

Application

The ISO 6983 standard defines a universal NC syntax.

Further information: "ISO example", Page 1132

On the TNC7 basic, you can program and execute NC programs using the supported ISO syntax elements.

Description of function

In connection with ISO programs, the TNC7 basic provides the following possibilities:

- Transferring files to the control

  Further information: "PC software for data transfer", Page 1875
- Programming ISO programs on the control

  Further information: "ISO syntax", Page 1135

  - In addition to the standardized ISO syntax, you can program HEIDENHAIN-specific cycles as G functions.

    Further information: "Cycles", Page 1154
  - Coding in Klartext syntax allows you to use some NC functions in ISO programs.

    Further information: "Klartext functions in ISO programming", Page 1156
- Testing of NC programs using Simulation mode

  Further information: "The Simulation Workspace", Page 1197
- Running NC programs

  Further information: "Program Run", Page 1637

Contents of an ISO program

An ISO program is structured as follows:

ISO syntax	Function
I	File type ISO programs have an ***.i** file name extension.
%NAME G71	Start and end of the program
G71	Unit of measure: mm
G70	Unit of measure: Inch
N10 **N20** **N30** ...	NC block numbers In the optional machine parameter **blockIncrement** (no. 105409), you define the increment between the block numbers.
N99999999	NC block number for the end of the program An NC program is incomplete without this NC block number. The control adds and updates the NC block numbers within the file automatically. The **Program** workspace exclusively shows successive numbers without taking the defined increment into account.
G01 X+0 Y+0 ...	NC functions

Further information: "Contents of an NC program", Page 199

Contents of an NC block

N110 G01 G90 X+10 Y+0 G41 F3000 M3

An NC block contains the following syntax elements:

ISO syntax	Function
G01	Start of syntax
G90	Absolute or incremental input
	Further information: "Absolute and incremental input", Page 1135
X+10 Y+0	Coordinates
	Further information: "Fundamentals of coordinate definitions", Page 292
G41	Tool radius compensation
	Further information: "Tool radius compensation", Page 1146
F3000	Feed rate
	Further information: "Feed rate", Page 1137
M3	Miscellaneous functions (M functions)
	Further information: "Miscellaneous Functions", Page 963

ISO example

Example task 1338459

					ID number			
Text:					Change No.	C000941-05		
					Phase:	Nicht-Serie		
⊕	Original drawing		**Platte**		Werkstoff: Material:			
	Scale	Format						
RoHS	1:1	A4	**Plate**					
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		Oberflächen nach ISO 1302 Surfaces as per ISO 1302	
					Oberflächenbehandlung: Surface treatment:			
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 DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany		Created	Responsible	Released	Version	Revision	Sheet	Page
		M-TS			**D1358459-00-A-01**			1
		05.08.2021			Document number			of 1

Example solution 1338459

% 1339889 G71	
N10 G30 G17 X+0 Y+0 Z-40	; Workpiece blank definition
N20 G31 X+100 Y+100 Z+0	; Workpiece blank definition
N30 T16 G17 S6500	; Tool call
N40 G00 G90 Z+250 G40 M3	; Clearance height in the tool axis
N50 G00 X-20 Y-20	; Pre-positioning in the machining plane
N60 G00 Z+5	; Pre-positioning in the tool axis
N70 G01 Z-5 F3000 M8	; Feed to working depth
N80 G01 X+5 Y+5 G41 F700	; First contour point
N90 G26 R8	; Approach function
N100 G01 Y+95	; Straight line
N110 G01 X+95	
N120 G24 R10	; Chamfer
N130 G01 Y+5	
N140 G24 R20	
N150 G01 X+5	
N160 G27 R8	; Departure function
N170 G01 X-20 Y-20 G40 F1000	; Clearance height in the machining plane
N180 G00 Z+250	; Clearance height in the tool axis
N190 T6 G17 S6500	; Tool call
N200 G00 G90 Z+250 G40 M3	
N210 G00 X+50 Y+50 M8	
N220 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**	
N230 G79		; Cycle call
N240 G00 Z+250 M30		
N99999999 % 1339889 G71		

Notes

- The **Insert NC function** window allows you add ISO syntax, too.

  Further information: "The Insert NC function window", Page 215

- You can call a Klartext program within an ISO program (e.g., to benefit from the possibilities of graphical programming).

  Further information: "Calling an NC program", Page 1143

  Further information: "Graphical programming", Page 1089

- You can call a Klartext program within an ISO program (e.g., to use NC functions that are available only for Klartext programming).

  Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 940

27.2 ISO syntax

27.2.1 Keys

You can use the following keys to insert ISO syntax:

Key	ISO syntax	Further information
TOOL CALL	Tool call **T**	Page 1136
TOOL DEF	Tool definition **G99**	Page 1137
L	Straight line **G01**	Page 1138
CHF	Chamfer **G24**	Page 1138
RND	Rounding arc **G25**	Page 1139
CC	Circular arc **G02**	Page 1140
C	Circular arc **G03**	Page 1140
CR	Circular arc **G05**	Page 1140
CT	Tangential arc **G06**	Page 1141
LBL SET	Label **G98**	Page 1142
LBL CALL	Subprogram call and program-section repeat **L**	Page 1143 Page 1143
STOP	Stop in the NC program **G38**	Page 1146

Absolute and incremental input

The control provides the following possibilities to enter dimensions:

Syntax	Meaning
G90	Absolute input always references an origin. For Cartesian coordinates, the origin is the datum, and for polar coordinates the origin is the pole and the angle reference axis.
G91 corresponds to the **I** Klartext syntax	Incremental input always references the previously programmed coordinates. For Cartesian coordinates, these are the values in the **X**, **Y**, and **Z** axes, and for polar coordinates, the values of the polar coordinate radius **R** and the polar coordinate angle **H**.

Tool axis

In some NC functions, you can select a tool axis in order, for example, to define the working plane.

⚙ The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

The control differentiates between the following tool axes:

Syntax	Working plane
G17 corresponds to the **Z** tool axis	**XY**, as well as **UV**, **XV**, **UY**
G18 corresponds to the **Y** tool axis	**ZX**, as well as **VW**, **YW**, **VZ**
G19 corresponds to the **X** tool axis	**YZ**, as well as **WU**, **ZU**, **WX**

Workpiece blank

Use the **G30** and **G31** NC functions to define a cuboid workpiece blank for simulation in the NC program.

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.

N10 G30 G17 X+0 Y+0 Z-40	; Define MIN point
N20 G31 X+100 Y+100 Z+0	; Define MAX point

G30 and **G31** correspond to the Klartext syntax **BLK FORM 0.1** and **BLK FORM 0.2**.
Further information: "Defining a workpiece blank with BLK FORM", Page 244
With **G17**, **G18**, and **G19**, you define the tool axis.
Further information: "Tool axis", Page 1136

With the Klartext syntax, you can additionally define the following workpiece blanks:
■ Cylindrical workpiece blank with **BLK FORM CYLINDER**
  Further information: "Cylindrical workpiece blank with BLK FORM CYLINDER", Page 247
■ Rotationally symmetric workpiece blank with **BLK FORM ROTATION**
  Further information: "Rotationally symmetric workpiece blank with BLK FORM ROTATION", Page 248
■ STL file as workpiece blank with **BLK FORM FILE**
  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 249

Tools

Tool call

With the **T** NC function, you call a tool in the NC program.
T corresponds to the **TOOL CALL** Klartext syntax.
Further information: "Tool call by TOOL CALL", Page 279
With **G17**, **G18**, and **G19**, you define the tool axis.
Further information: "Tool axis", Page 1136

Cutting data

Spindle speed

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

N110 T1 G17 S(VC = 200)	; Tool call with constant cutting speed

Further information: "Spindle speed S", Page 283

Feed rate

The feed rate for linear axes is defined in millimeters per minute (mm/min).

In inch programs, the feed rate must be defined in 1/10 inch/min.

The feed rate for rotary axes is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Further information: "Feed rate F", Page 284

Tool definition

With the **G99** NC function, you can define the dimensions/allowance of a tool.

> ⚙ Refer to your machine manual.
> A tool definition created with **G99** is a machine-dependent function.
> HEIDENHAIN recommends using tool management for the definition of tools instead of **G99**!
> **Further information:** "Tool management ", Page 269

110 G99 T3 L+10 R+5	; Define tool

G99 corresponds to the **TOOL DEF** Klartext syntax.

Further information: "Tool pre-selection by TOOL DEF", Page 286

Tool pre-selection

When you use the **G51** NC function, the control prepares a tool in the magazine, thus reducing the tool-change time.

> ⚙ Refer to your machine manual.
> A tool pre-selection defined with **G99** is a machine-dependent function.

110 G51 T3	; Tool pre-selection

G51 corresponds to the **TOOL DEF** Klartext syntax.

Further information: "Tool pre-selection by TOOL DEF", Page 286

Path functions

Straight line

Cartesian coordinates

With the **G00** and **G01** NC functions, you program a straight movement in rapid traverse or with a machining feed rate in any desired direction.

N110 G00 Z+100 M3	; Straight line at rapid traverse
N120 G01 X+20 Y-15 F200	; Straight line at machining feed rate

If the feed rate was programmed using a numerical value, it is active only up to the NC block in which a new feed rate is programmed. **G00** is active only for the NC block in which it was programmed. When the NC block programmed with **G00** has been executed, the feed rate programmed most recently with a numerical value becomes active again.

> **ℹ** Make sure to program rapid traverse movements exclusively with the **G00** NC function instead of very high numerical values. This is the only way to ensure that rapid traverse is active on a block-by-block basis and that you can control rapid traverse independently of the machining feed rate.

G00 and **G01** correspond to the **L** Klartext syntax with **FMAX** and **F**.

Further information: "Straight line L", Page 300

Polar coordinates

With the **G10** and **G11** NC functions, you program a straight movement in rapid traverse or with a machining feed rate in any desired direction.

N110 I+0 J+0	; Pole
N120 G10 R+10 H+10	; Straight line at rapid traverse
N130 G11 R+50 H+50 F200	; Straight line at machining feed rate

The polar coordinate radius **R** corresponds to the **PR** Klartext syntax.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

G10 and **G11** correspond to the **LP** Klartext syntax with **FMAX** and **F**.

Further information: "Straight line LP", Page 320

Chamfer

With the **G24** NC function, you can insert a chamfer between two straight lines. The chamfer size references the point of intersection you are programming using the straight line.

N110 G01 X+40 Y+5	; Straight line at machining feed rate
N120 G24 R12	; Chamfer at machining feed rate
N130 G01 X+5 Y+0	; Straight line at machining feed rate

The value following the **R** syntax element corresponds to the chamfer size.

G24 corresponds to the **CHF** Klartext syntax.

Further information: "Chamfer CHF", Page 302

Rounding arc

With the **G25** NC function, you can insert a rounding arc between two straight lines. The rounding arc references the point of intersection you are programming using the straight line.

N110 G01 X+40 Y+25	; Straight line at machining feed rate
N120 G25 R5	; Rounding arc at machining feed rate
N130 G01 X+10 Y+5	; Straight line at machining feed rate

G25 corresponds to the **RND** Klartext syntax.

The value following the **R** syntax element corresponds to the radius of the rounding arc.

Further information: "Rounding RND", Page 304

Circle center

Cartesian coordinates

With the **I**, **J**, and **K** or **G29** NC functions, you define the circle center.

| **N110 I+25 J+25** | ; Circle center in the XY plane |

| **N110 G00 X+25 Y+25** | ; Pre-positioning on a straight line |
| **N120 G29** | ; Circle center at the last position |

- **I**, **J**, and **K**

  The circle center is defined in this NC block.
- **G29**

  The control assumes the most recently programmed position as the circle center.

I, **J**, and **K** or **G29** correspond to the **CC** Klartext syntax with or without axis values.

Further information: "Circle center point CC", Page 306

> **i** With **I** and **J**, you define the circle center in the **X** and **Y** axes. In order to define the **Z** axis, program **K**.
>
> **Further information:** "Circular path in another plane", Page 317

Polar coordinates

With the **I**, **J**, and **K** or **G29** NC functions, you define a pole. All polar coordinates reference the pole.

| **N110 I+25 J+25** | ; Pole |

- **I**, **J**, and **K**

  The pole is defined in this NC block.
- **G29**

  The control takes over the most recently programmed position as the pole.

I, **J**, and **K** or **G29** correspond to the **CC** Klartext syntax with or without axis values.

Further information: "Polar coordinate datum at pole CC", Page 319

Circular arc with center

Cartesian coordinates

With the **G02**, **G03**, and **G05** NC functions, you program a circular path around a circle center.

N110 I+25 J+25	; Circle center
N120 G03 X+45 Y+25	; Circular path around circle center

- **G02**

  Circular path in clockwise direction, corresponds to the **C** Klartext syntax with **DR-**.

- **G03**

  Circular path in counterclockwise direction, corresponds to the **C** Klartext syntax with **DR+**.

- **G05**

  Circular path without direction of rotation, corresponds to the **C** Klartext syntax without **DR**.

  The control uses the most recently programmed direction of rotation.

Further information: "Circular path C ", Page 308

> **i**   When you program a radius **R**, there is no need to define a circle center.
> **Further information:** "Circular path with a defined radius", Page 1141

Polar coordinates

With the **G12**, **G13**, and **G15** NC functions, you program a circular path around a defined pole.

N110 I+25 J+25	; Pole
N120 G13 H+180	; Circular path around pole

- **G12**

  Circular path in clockwise direction, corresponds to the **CP** Klartext syntax with **DR-**.

- **G13**

  Circular path in counterclockwise direction, corresponds to the **CP** Klartext syntax with **DR+**.

- **G15**

  Circular path without direction of rotation; corresponds to the **CP** Klartext syntax without **DR**.

  The control uses the most recently programmed direction of rotation.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

Further information: "Circular path CP around pole CC", Page 323

Circular path with a defined radius

Cartesian coordinates

With the **G02**, **G03**, and **G05** NC functions, you program a circular path with a defined radius. If you are programming a radius, no circle center is required.

N110 G03 X+70 Y+40 R+20	; Circular path with a defined radius

- **G02**

  Circular path in clockwise direction, corresponds to the **CR** Klartext syntax with **DR-**.
- **G03**

  Circular path in counterclockwise direction, corresponds to the **CR** Klartext syntax with **DR+**.
- **G05**

  Circular path without direction of rotation; corresponds to the **CR** Klartext syntax without **DR**.

  The control uses the most recently programmed direction of rotation.

Further information: "Circular path CR", Page 310

Circular arc with a tangential transition

Cartesian coordinates

With the **G06** NC function, you program a circular path with a tangential transition to the previous path function.

N110 G01 X+25 Y+30 F300	; Straight line
N120 G06 X+45 Y+20	; Circular path with tangential transition

G06 corresponds to the **CT** Klartext syntax.

Further information: "Circular path CT", Page 313

Polar coordinates

With the **G16** NC function, you program a circular path with a tangential transition to the previous path function.

N110 G01 G42 X+0 Y+35 F300	; Straight line
N120 I+40 J+35	; Pole
N130 G16 R+25 H+120	; Circular path with tangential transition

The polar coordinate radius **R** corresponds to the **PR** Klartext syntax.

The polar coordinate angle **H** corresponds to the **PA** Klartext syntax.

G16 corresponds to the **CTP** Klartext syntax.

Further information: "Circular path CTP", Page 325

Contour approach and departure

With the **G26** and **G27** NC functions, you can approach or depart the contour smoothly using a circle segment.

N110 G01 G40 G90 X-30 Y+50	; Starting point
N120 G01 G41 X+0 Y+50 F350	; First contour point
N130 G26 R5	; Tangential approach
* - ...	
N210 G27 R5	; Tangential exit
N220 G00 G40 X-30 Y+50	; End point

HEIDENHAIN recommends the use of the more powerful **APPR** and **DEP** NC functions. In some cases, these NC functions combine multiple NC blocks for approaching and departing the contour.

G41 and **G42** correspond to the **RL** and **RR** Klartext syntax.

Further information: "Approach and departure functions with Cartesian coordinates", Page 333

You can also use polar coordinates when programming the **APPR** and **DEP** NC functions.

Further information: "Approach and departure functions with polar coordinates", Page 347

Programming techniques

Subprograms and program-section repeats

Programming techniques are useful in structuring your NC program and avoiding unnecessary repeats. By using subprograms, you need to define machining positions for multiple tools only once, for example. Program-section repeats, on the other hand, help you avoid multiple programming of identical, successive NC blocks or program sequences. By combining and nesting these two programming techniques, you can keep your NC programs rather short and restrict changes to a few central program locations.

Further information: "Subprograms and program section repeats with the label LBL", Page 360

Defining labels

With the **G98** NC function, you define a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by its number or name. If a number or a name exists twice in an NC program, the control shows a warning before the NC block.

If you define a label after **M30** or **M2**, it corresponds to a subprogram. Subprograms must always be concluded with a **G98 L0**. This number is the only one which may exist any number of times in the NC program.

N110 G98 L1	; Start of subprogram defined by a number
N120 G00 Z+100	, Retract at rapid traverse
N130 G98 L0	; End of subprogram

N110 G98 L "UP"	; Start of subprogram defined by a name

G98 L corresponds to the **LBL** Klartext syntax.

Further information: "Defining a label with LBL SET", Page 360

Calling a subprogram

With the **L** NC function, you call a subprogram programmed after **M30** or **M2**.

When the control reads the **L** NC function, it will jump to the defined label and continue execution of the NC program from this NC block. When the control reads **G98 L0**, it will jump back to the next NC block after the call with **L**.

| N110 L1 | ; Call subprogram |

L without **G98** corresponds to the **CALL LBL** Klartext syntax.

Further information: "Calling a label with CALL LBL", Page 361

> **i** In order to define a certain number of desired repetitions (e.g., **L1.3**), program a program-section repeat.
> **Further information:** "Program-section repeat", Page 1143

Program-section repeat

Program-section repeats allow you to have a particular program section executed any number of times. The program section must start with a **G98 L** label definition and end with **L**. With the numeral after the decimal point, you can define optionally how often you want the control to repeat this program section.

| N110 L1.2 | ; Call label 1 twice |

L without **98** and the numeral after the decimal point correspond to the **CALL LBL REP** Klartext syntax.

Further information: "Program-section repeats", Page 363

Selection functions

Further information: "Selection functions", Page 364

Calling an NC program

With the **%** NC function, you can call another, separate NC program from within an NC program.

| N110 %TNC:\nc_prog\reset.i | ; Call NC program |

% corresponds to the **CALL PGM** Klartext syntax.

Further information: "Call the NC program with CALL PGM", Page 364

Activating a datum table in the NC program

With the **%:TAB:** NC function, you can activate a datum table from within an NC program.

| N110 %:TAB: "TNC:\table\zeroshift.d" | ; Activate datum table |

%:TAB corresponds to the **SEL TABLE** Klartext syntax.

Further information: "Activating the datum table in the NC program", Page 716

Selecting a point table

With the **%:PAT:** NC function, you can activate a point table from within an NC program.

N110 %:PAT: "TNC:\nc_prog \positions.pnt"	; Activate point table

%:PAT corresponds to the **SEL PATTERN** Klartext syntax.

Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 393

Selecting an NC program with contour definitions

With the **%:CNT:** NC function, you can select another NC program with a contour definition from within an NC program.

N110 %:PAT: "TNC:\nc_prog\contour.h"	; Select NC program with contour definition

Further information: "Graphical programming", Page 1089

%:CNT corresponds to the **SEL CONTOUR** Klartext syntax.

Further information: "Selecting an NC program with contour definition", Page 386

Selecting and calling an NC program

With the **%:PGM:** NC function, you can select another, separate NC program. With the **%<>%** NC function, you call the selected NC program at a different location in the active NC program.

N110 %:PGM: "TNC:\nc_prog\reset.i"	; Select NC program
* - ...	
N210 %<>%	; Call the selected NC program

%:PGM: and **%<>%** correspond to the **SEL PGM** and **CALL SELECTED PGM** Klartext syntax.

Further information: "Call the NC program with CALL PGM", Page 364

Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 366

Defining an NC program as a cycle

With the **G: :** NC function, you can define another NC program as a machining cycle from within an NC program.

N110 G: : "TNC:\nc_prog\cycle.i"	; Define NC program as a machining cycle

G: : corresponds to the **SEL CYCLE** Klartext syntax.

Further information: "Defining and calling an NC program as cycle", Page 229

Cycle call

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 machining cycle that was defined last in the NC program.

The control provides the following options for calling a cycle:

Syntax	Meaning
G79 corresponds to the **CYCL CALL** Klartext syntax	The control calls the most recently programmed machining cycle at the last programmed position.
G79 PAT corresponds to the **CYCL CALL PAT** Klartext syntax	The control calls the most recently programmed machining cycle at all positions you have defined in a point table.
G79|G01 corresponds to the **CYCL CALL POS** Klartext syntax	The control calls the most recently programmed machining cycle at the position you defined in the NC block with **G79|G01**.
M89 and **M99**	With **M99**, the control executes the most recently programmed machining cycle at the most recently programmed position.
	With **M89**, the control executes the most recently programmed machining cycle after each positioning block until it reads **M99**.

N110 G79 M3	; Call cycle

N110 G79 PAT F200 M3	; Call cycle at all positions in the point table

N110 G79|G01 G90 X+0 X+25	; Call cycle at the defined position

N110 G01 X+0 X+25 M89	; Call cycle at the defined position and for each new positioning block
N120 G01 X+25 Y+25	
N130 G01 X+50 Y+25 M99	; Call cycle for the last time at the defined position

Further information: "Calling cycles", Page 227

Tool radius compensation

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.

An NC block can contain the following tool radius compensations:

Syntax	Meaning
G40 corresponds to the **R0** Klartext syntax	Reset an active tool radius compensation, positioning based on the tool center point
G41 corresponds to the **RL** Klartext syntax	Tool radius compensation, on the left of the contour
G42 corresponds to the **RR** Klartext syntax	Tool radius compensation, on the right of the contour

Further information: "Tool radius compensation", Page 798

Miscellaneous functions (M functions)

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Further information: "Miscellaneous Functions", Page 963

G38 corresponds to the **STOP** Klartext syntax.

Further information: "Miscellaneous functions M and the STOP function ", Page 964

Programming variables

The control provides the following options for programming variables in
ISO programs:

Function group	Further information
Basic arithmetic operations	Page 1148
Trigonometric functions	Page 1149
Circle calculations	Page 1150
Jump commands	Page 1151
Special functions	Page 1153
String functions	Corresponds to the Klartext syntax Page 1048
Counters	Corresponds to the Klartext syntax Page 1058
Calculations using formulas	Corresponds to the Klartext syntax Page 1043
Function for the definition of complex contours	Corresponds to the Klartext syntax Page 383

The control distinguishes between the **Q**, **QL**, **QR**, and **QS** variable types (parameter
types).
Further information: "Variable Programming", Page 1005

> **i** Not all NC functions for programming variables are available in
> ISO programs (e.g., accessing tables with SQL statements).
> **Further information:** "Table access with SQL statements", Page 1066

Basic arithmetic operations

With the **D01** through **D05** functions, you can calculate values within your NC program. If you want to calculate with variables, you need to assign an initial value to each variable by means of the **D00** function.

The control provides the following functions:

Syntax	Meaning
D00	Assignment Assign a value or the **Undefined** status
D01	Addition Calculate and assign the sum of two values
D02	Subtraction Calculate and assign the difference of two values.
D03	Multiplication Calculate and assign the product of two values.
D04	Division Calculate and assign the quotient of two values Restriction: You cannot divide by 0
D05	Square root Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

N110 D00 Q5 P01 +60	; Assignment Q5 = 60
N110 D01 Q1 P01 -Q2 P02 -5	; Addition Q1 = −Q2+(−5)
N110 D02 Q1 P01 +10 P02 +5	; Subtraction Q1 = +10−(+5)
N110 D03 Q2 P01 +3 P02 +3	; Multiplication Q2 = 3*3
N110 D04 Q4 P01 +8 P02 +Q2	; Division Q4 = 8/Q2
N110 D05 Q20 P01 4	; Square root Q20 =√4

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Basic arithmetic folder", Page 1020

> **ℹ** HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
>
> **Further information:** "Formulas in the NC program", Page 1043

Trigonometric functions

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

The control provides the following functions:

Syntax	Meaning
D06	Sine
	Calculate and assign the sine of an angle in degrees
D07	Cosine
	Calculate and assign the cosine of an angle in degrees
D08	Root of the sum of squares
	Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
D13	Angle
	Calculate and assign the angle from the opposite side and the adjacent side using arctan or from the sine and cosine of the angle (0 < angle < 360°)

N110 D06 Q20 P01 -Q5	; Sine, Q20 = sin(−Q5)
N110 D07 Q21 P01 -Q5	; Cosine, Q21 = cos(−Q5)
N110 D08 Q10 P01 +5 P02 +4	; Root of the sum of squares, Q10 = $\sqrt{(5^2+4^2)}$
N110 D13 Q20 P01 +10 P02 -Q1	; Angle, Q20 = arctan(25/−Q1)

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Trigonometric functions folder", Page 1022

> ℹ️ HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
>
> **Further information:** "Formulas in the NC program", Page 1043

Circle calculation

These functions allow you to calculate the center of a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

The control provides the following functions:

Syntax	Meaning
D23	Circle data from three points on the circle
	The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.
D24	Circle data from four points on the circle
	The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.

N110 D23 Q20 P01 Q30	; Circle data from three points on the circle

N110 D24 Q20 P01 Q30	; Circle data from four points on the circle

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Circle calculation folder", Page 1024

Jump commands

In if-then decisions, the control compares a variable or fixed value with another variable or fixed value. If the condition is fulfilled, the control jumps to the label programmed for the condition.

If the condition is not fulfilled, the control continues with the next NC block.

The control provides the following functions:

Syntax	Meaning
D09	Jump if equal
	If both values are equal, the control jumps to the defined label.
	Jump if undefined
	If the variable is undefined, the control jumps to the defined label.
	Jump if defined
	If the variable is defined, the control jumps to the defined label.
D10	Jump if not equal
	If both values are not equal, the control jumps to the defined label.
D11	Jump if greater than
	If the first value is greater than the second one, the control jumps to the defined label.
D12	Jump if less than
	If the first value is less than the second one, the control jumps to the defined label.

N110 D09 P01 +Q1 P02 +Q3 P03 "LBL"	; Jump if equal
N110 D09 P01 +Q1 IS UNDEFINED P03 "LBL"	; Jump if undefined
N110 D09 P01 +Q1 IS DEFINED P03 "LBL"	; Jump if defined
N110 D10 P01 +10 P02 -Q5 P03 10	; Jump if not equal
N110 D11 P01 +Q1 P02 +10 P03 QS5	; Jump if greater than
N110 D12 P01 +Q5 P02 +0 P03 "LBL"	; Jump if less than

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information:

Functions for freely definable tables

You can open any free definable table and subsequently write to it or read from it.

The control provides the following functions:

Syntax	Meaning
D26	Open a freely definable table
	Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1039
D27	Write to a freely definable table
	Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1039
D28	Read from a freely definable table
	Further information: "Reading a freely definable table with FN 28: TABREAD", Page 1041

N110 D26 TNC:\DIR1\TAB1.TAB	; Open a freely definable table

N110 Q5 = 3.75	; Define the value for the **Radius** column
N120 Q6 = -5	; Define the value for the **Depth** column
N130 Q7 = 7,5	; Define the value for the **D** column
N140 D27 P01 5/"Radius,Depth,D" = Q5	; Write defined values to the table

N110 D28 Q10 = 6/"X,Y,D"*	; Read numerical values from the **X**, **Y**, and **D** columns
N120 D28 QS1 = 6/"DOC"*	; Read the alphanumeric value from the **DOC** column

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Special functions

The control provides the following functions:

Syntax	Meaning
D14	Output error messages
	Further information: "Output error messages with FN 14: ERROR", Page 1027
	Further information: "Preassigned error numbers for FN 14: ERROR", Page 1972
D16	Output formatted texts
	Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1028
D18	Read system data
	Further information: "Read system data with FN 18: SYSREAD", Page 1035
	Further information: "System data", Page 1978
D19	Transfer values to the PLC
	Further information: "Special functions defining the machine behavior", Page 1971
D20	Synchronize NC and PLC
	Further information: "Special functions defining the machine behavior", Page 1971
D29	Transfer values to the PLC
	Further information: "Special functions defining the machine behavior", Page 1971
D37	Create user-defined cycles
	Further information: "Special functions defining the machine behavior", Page 1971
D38	Send information from the NC program
	Further information: "Sending information from the NC program with FN 38: SEND", Page 1036

N110 D14 P01 1000	; Output error message no. 1000

N110 D16 P01 F-PRINT TNC:\mask.a / TNC: \Prot1.txt	; Display the output file with **D16** on the control screen

N110 D18 Q25 ID210 NR4 IDX3	; Save the active dimension factor of the Z axis in **Q25**

N110 D38 /"Q-Parameter Q1: %F Q23: %F" P02 +Q1 P02 +Q23	; Write the values of **Q1** and **Q23** to the log

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

> ### *NOTICE*
>
> **Danger of collision!**
>
> Changes to the PLC can result in undesired behavior and serious errors (e.g., the control might become inoperable). For this reason, access to the PLC is password-protected. The functions **D19**, **D20**, **D29**, and **D37** enable HEIDENHAIN, the machine manufacturer, and suppliers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is a danger of collision during the execution of these functions and during the subsequent machining operations!
>
> ▶ Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
> ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

27.3 Cycles

Fundamentals

In ISO programs, you can use selected cycles with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

The numbers of the Klartext cycles correspond to the numbers of the G functions. There are exceptions for earlier cycles that have numbers below **200**. In these cases, the corresponding G function number is mentioned in the cycle description.

Further information: "Available cycle groups", Page 237

The following cycles are not available in ISO programs:

- Cycle **1 POLAR PRESET**
- Cycle **3 MEASURING**
- Cycle **4 MEASURING IN 3-D**
- Cycle **26 AXIS-SPECIFIC SCALING**

HEIDENHAIN recommends using the more powerful **PLANE** functions instead of Cycle **G80 WORKING PLANE**. With the **PLANE** functions, you can choose freely between axis or spatial angles for programming.

Further information: "PLANE SPATIAL", Page 742

Datum shift

With the **G53** or **G54** NC functions, you can program datum shifts. **G54** shifts the workpiece datum to the coordinates you define directly within this function. **G53** uses coordinate values from a datum table. A datum shift allows machining operations to be repeated at any locations on the workpiece.

N110 G54 X+0 Y+50	; Shift the workpiece datum to the defined coordinates

N110 G53 P01 10	; Shift the workpiece datum to the coordinates of table row 10

To reset a datum shift:

- Define the value **0** for each axis in function **G54**
- In function **G53**, select a table row where all columns have the value **0**

The control displays the following information in the **Status** workspace:

- Name and path of the active datum table
- Active datum number
- Comment from the **DOC** column of the active datum number

Notes

> ⚙ In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

- Datums from a datum table always reference the current workpiece preset.
- Before shifting the workpiece datum by means of a datum table, you need to activate the datum table with **%:TAB:**

  Further information: "Activating a datum table in the NC program", Page 1143
- If you do not use **%:TAB:**, you have to activate the datum table manually.

  Further information: "Activating the datum table manually", Page 715

27.4 Klartext functions in ISO programming

Fundamentals

In ISO programs, you can use selected NC functions with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

For more information about programming, refer to the respective chapters describing the individual NC functions.

The following NC functions are available only in Klartext programs:

- Pattern definitions with **PATTERN DEF**

  Further information: "Pattern definition with PATTERN DEF", Page 394

- NC functions for coordinate transformations: **TRANS DATUM**, **TRANS MIRROR**, **TRANS ROTATION**, and **TRANS SCALE**

  Further information: "NC functions for coordinate transformation", Page 726

- File functions: **FUNCTION FILE** and **OPEN FILE**

  Further information: "Programmable file functions", Page 842

- Functions for machining with parallel axes: **PARAXCOMP** and **PARAXMODE**

  Further information: "Working with the parallel axes U, V and W", Page 933

- Programs that use normal vectors

  Further information: "CAM-generated NC programs", Page 947

- Table access with SQL statements

  Further information: "Table access with SQL statements", Page 1066

- Changing kinematics with **WRITE KINEMATICS**

28

User aids

28.1 The Help workspace

Application

In the **Help** workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid **TNCguide**.

Related topics

- The **Help** application
  Further information: "The Help application", Page 67
- User's Manual as the **TNCguide** integrated product aid
  Further information: "User's Manual as integrated product aid: TNCguide", Page 66

Description of function

The **Help** workspace can be selected in the **Editor** operating mode and in the **MDI** application.

Further information: "The Editor operating mode", Page 202

Further information: "The MDI Application ", Page 1221

While the **Help** workspace is active, the control displays the help graphic there and not in a pop-up window.

Further information: "Help graphic", Page 206
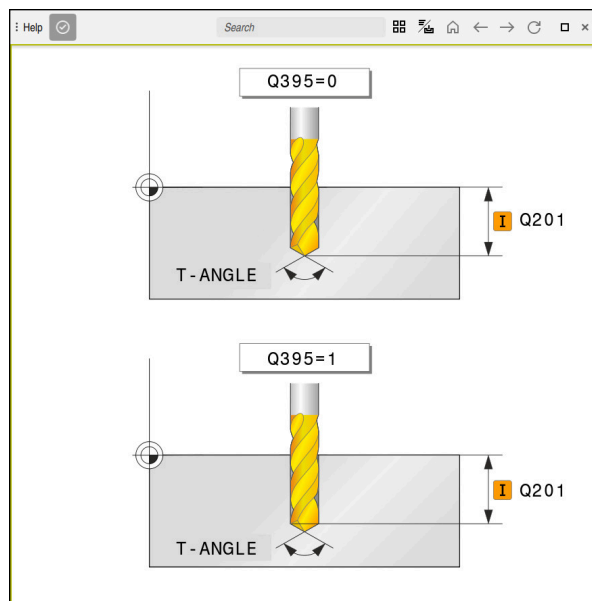
The **Help** workspace with a help graphic of a cycle parameter

When the **Help** workspace is active, the control can display the integrated **TNCguide** product aid.

Further information: "User's Manual as integrated product aid: TNCguide", Page 66
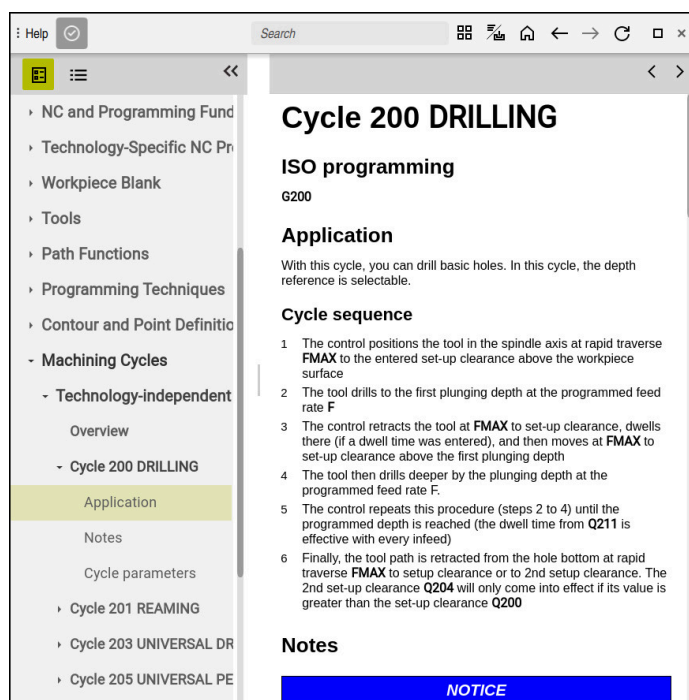
The **Help** workspace with **TNCguide** open

Icons

The following icons are shown in the **Help** workspace:

Icon	Meaning
⊘	Open or close the **Search results** column
	Further information: "Search in TNCguide", Page 69
⊞	**Open Home page**
	The start page displays all available documentation. Select the desired documentation using 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: "Icons", Page 68
⅔⧌	Open **TNCguide** or the **Help Graphic**
	The control toggles between **TNCguide** and the **Help Graphic**. The control will only display a Help Graphic if you edit an NC block for which an associated Help Graphic exists.
⌂	Open **TNCguide** in the **Help** application
	The control opens **TNCguide** at the current position.
	Further information: "The Help application", Page 67
← →	**Navigate**
	Navigate between the contents opened recently
⟳	**Refresh**

TNCguide has additional icons.

Further information: "User's Manual as integrated product aid: TNCguide", Page 66

28.2 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
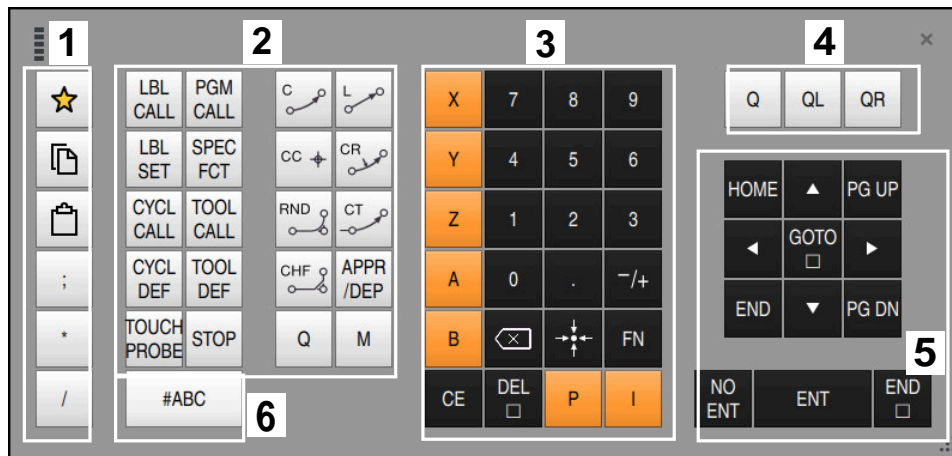
Virtual keyboard in NC input mode

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

> **i** 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**

Text input areas

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

Formula input areas
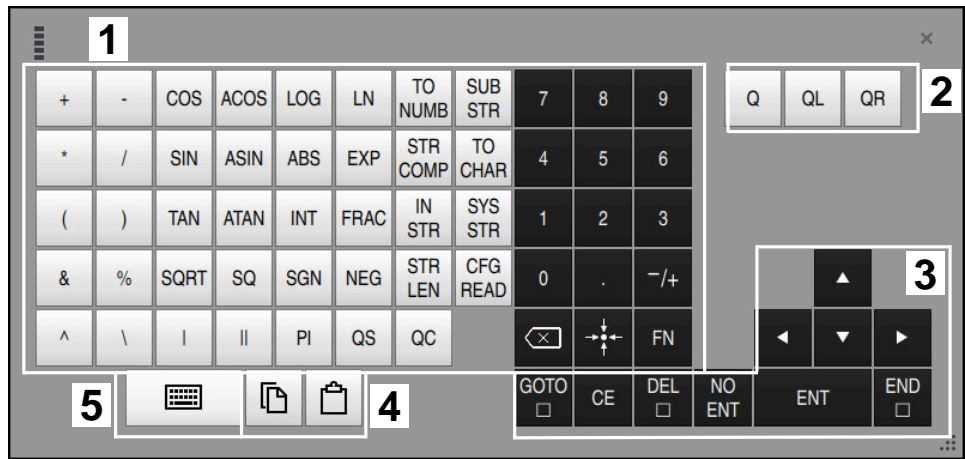
Virtual keyboard in formula input mode

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

28.2.1 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.

28.3 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.

28.3.1 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 1649

Notes

- Instead of the **GOTO** button, you can also use the **CTRL + G** shortcut.
- If the control in the action bar shows an icon for selection, you can open the selection window with **GOTO**.

28.4 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.

Comments that include line breaks can only be edited in the Text editor mode or in the **Form** column.

Further information: "Using the Program workspace", Page 211

28.4.1 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

28.4.2 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

28.4.3 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

| ; Comment Off/On | ▶ 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.

28.5 Hiding NC blocks

Application

Use **/** or the **Skip block Off/On** button to hide NC blocks.

By hiding NC blocks, you can skip the corresponding NC blocks in the program run.

Related topics

■ The **Program Run** operating mode
Further information: "The Program Run operating mode", Page 1638

Description of function

If you mark an NC block with a **/** character, then the NC block is hidden. If you activate the **Skip block** toggle switch in the **Program Run** operating mode or in the **MDI** application, the control skips the NC block during execution.

If the toggle switch is active, then the control dims the NC blocks to be skipped.

Further information: "Icons and buttons", Page 1640

28.5.1 Hiding or showing NC blocks

To hide or show an NC block:

▶ Select the desired NC block
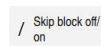
| / Skip block off/on | ▶ Select **Skip block Off/On** |

> The control adds a **/** character before the NC block.
> If the NC block is already hidden, the control removes the **/** character.

28.6 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

■ The **Structure** column of the **Program** workspace

Further information: "The Structure column in the Program workspace", Page 1166

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 structuring items in the **Structure** column.

Further information: "The Structure column in the Program workspace", Page 1166

28.6.1 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

28.7 The Structure column in the Program workspace

Application

When you open an NC program, the control searches the NC program for structuring items and displays these structure elements in the **Structure** column. The structuring items act like links and thus allow fast navigation in the NC program.

Related topics
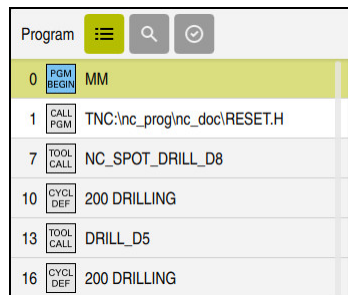
■ The **Program** workspace, defining contents of the **Structure** column

Further information: "Settings in the Program workspace", Page 206

■ Inserting structure items manually

Further information: "Structuring of NC programs", Page 1166

Description of function

The **Structure** column with automatically created structuring items

When you open an NC program, the control automatically creates the structure.

In the **Program settings** window, you define which structuring items the control displays in the structure. The **PGM BEGIN** and **PGM END** structuring items cannot be hidden.

Further information: "Settings in the Program workspace", Page 206

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
BEGIN PGM	**BEGIN PGM**	Unit of measurement of the NC program **MM** or **INCH**
TOOL CALL	**TOOL CALL**	■ Name or number of the tool, if applicable ■ Index of the tool, if applicable ■ Comment, if applicable
*	*** Structure block**	■ Entered string, if applicable ■ Comment, if applicable
LBL SET	**LBL SET**	■ Name or number of the label ■ Comment, if applicable
LBL SET	**LBL 0**	■ Number of the label ■ Comment, if applicable
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, if applicable ■ Comment, if applicable
MON STOP	**MONITORING SECTION STOP**	Comment, if applicable
CALL PGM	■ **CALL PGM** ■ **CALL SELECTED PGM**	■ Path of the called NC program (e.g., **TNC:\Safe.h**), if applicable ■ Comment, if applicable
SEL PGM	■ Cycle **12.1 PGM** ■ **SEL PGM**	■ Path of the NC program (e.g., **TNC:\Safe.h**) ■ Comment, if applicable

Icon	Syntax	Information
SPEC FCT	**FUNCTION MODE**	■ Selected machining mode (possibilities: **MILL**, and **SET**) ■ Selected kinematics, if applicable ■ Comment, if applicable
M2 M30	**M2** or **M30**	Comment, if applicable
M1	**M1**	Comment, if applicable
STOP M0	**STOP** or **M0**	Comment, if applicable
APPR	**APPR**	■ Selected approach function ■ Comment, if applicable
DEP	**DEP**	■ Selected departure function ■ Comment, if applicable
END PGM	**PGM END**	No additional information

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.

Further information: "Navigation path in the Program workspace", Page 1646

> ⓘ The control displays comments as separate NC blocks, rather than including them in the structure. These NC blocks start with the semicolon **;**character.
> **Further information:** "Adding comments", Page 1164

28.7.1 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.

28.7.2 Marking NC blocks using the structure

To mark NC blocks using the structure:

▶ Open an NC program

☰   ▶ Open the **Structure** column

    ▶ Hold or right-click the structuring item

    > The control positions the cursor on the corresponding NC block in the NC program.

    > The control opens the context menu.
      Further information: "Context menu", Page 1174

    ▶ Select **Mark**

    > The control displays check boxes next to the structuring items in the **Structure** column.

    > The control marks the NC block in the NC program.

    ▶ Enable additional check boxes, if required

    > The control marks all structuring items between the two selected structuring items as well as the associated NC blocks.

> **i** Instead of the context menu, you can use the **CTRL + SPACE** shortcut.

Notes

- In the case of long NC programs, generating the structure view may take longer than loading the NC program itself. Even if the structure view has not been fully generated, you can already work 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. If the NC program is open, the control switches to the corresponding tab.

28.8 The Search column in the Program workspace

Application

In the **Search** column, you can search the NC program for any character strings, such as 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 213

Description of function
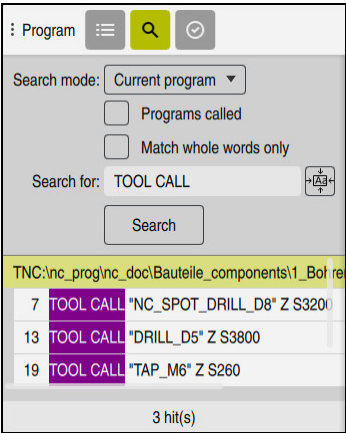
The **Search** column in the **Program** workspace

The control provides the full range of functions in the **Editor** operating mode only. In the **MDI** application, you can search only the active NC program. The **Search and replace** mode is not available in the **Program Run** operating mode.

The control provides the following functions, icons and buttons in the **Search** column:

Area	Function

Search mode:	■ **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, such as syntax elements
	  Further information: "Search and replace mode", Page 1171

Match whole words only	If you select the check box, the control only displays exact matches. This means that if you search for **Z+10**, for example, the control ignores **Z+100**. The check box is available in every mode.
Search for:	In the input area, you define the search term. If you have not yet entered any characters, the control suggests the last six search terms for selection. The search is not case-sensitive.
→⬚Aa⬚←	Use the **Apply selection** icon to transfer the currently select-ed syntax element to the input area. If the selected NC block is not edited, the control accepts the syntax initiator.
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, such as individual 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 check boxes 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 contents in the NC program with the contents 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.

28.8.1 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 mode:** 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**)

[Find next]

▶ Select **Find next**
> The control closes previously called NC programs, if any had been called, and highlights the first result in the main program in purple.

[Replace]

▶ Select **Replace**
> The control performs a syntax check and replaces the content if the check is successful.
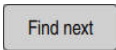
Notes

- 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.
- If you have not entered a value for **Replace with:**, the control deletes the search value.

28.9 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. Some NC blocks, particularly those consisting of cycles, can contain several lines within one block number.

  Further information: "Contents of an NC program", Page 199

Description of function
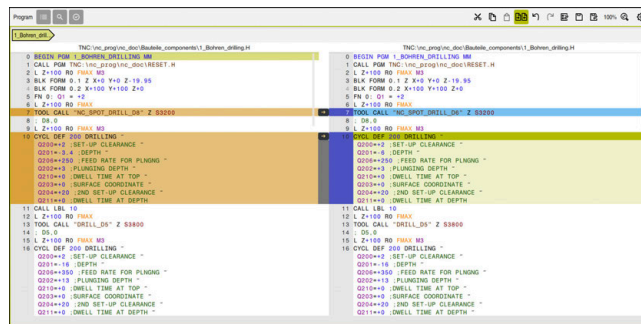
Program comparison of two NC programs

You can use the program comparison in the **Editor** operating mode in the **Program** workspace only.

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.

28.9.1 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.
- If you mark multiple NC blocks in the comparison program, you can apply those NC blocks simultaneously. If you mark multiple NC blocks in the active NC program, you can overwrite those NC blocks simultaneously.
  Further information: "Context menu", Page 1174

28.10 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
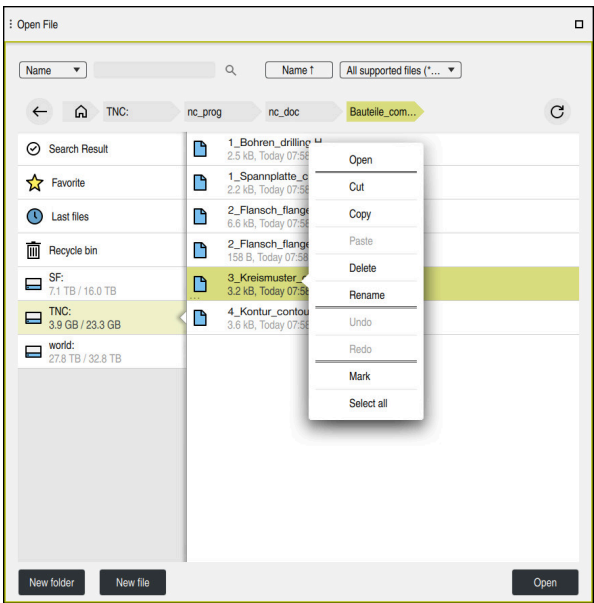
Context menu in the **Open File** workspace

Depending on the selected workspace and operating mode, the context menu provides the following functions:

- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Undo**
- **Redo**
- **Mark**
- **Select all**

> **i** 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 107

Key or keyboard shortcut	Meaning
CTRL + SPACE	Mark the selected line
SHIFT + UP	Additionally mark a line above it
SHIFT + DOWN	Additionally mark a line below it
SHIFT + PG UP	Mark from the cursor position to the beginning of the page Not available in the **Tables** operating mode
SHIFT + PG DN	Mark from the cursor position to the end of the page Not available in the **Tables** operating mode
SHIFT + HOME	Mark from the cursor position to the first row Not available in the **Tables** operating mode
SHIFT + END	Mark from the cursor position to the last row Not available in the **Tables** operating mode
ESC	Cancel marking

> **i** These 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 offers the following additional 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 1174

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.

In the **Tables** operating mode, the context menu provides some functions applicable both for cells and rows.

To cut or copy an entire table row, the control provides the following functions in the action bar:

- **Overwrite**

  The control inserts the row instead of the currently selected table row.

- **Append**

  The control appends the row at the end of the table.

> **ℹ** If the clipboard of the **Tool management** application contains indexed tools only, the control will create the rows as indices of the currently selected tool.

- **Cancel**

Further information: "The Tables operating mode", Page 1666

Context menu in the Job list workspace
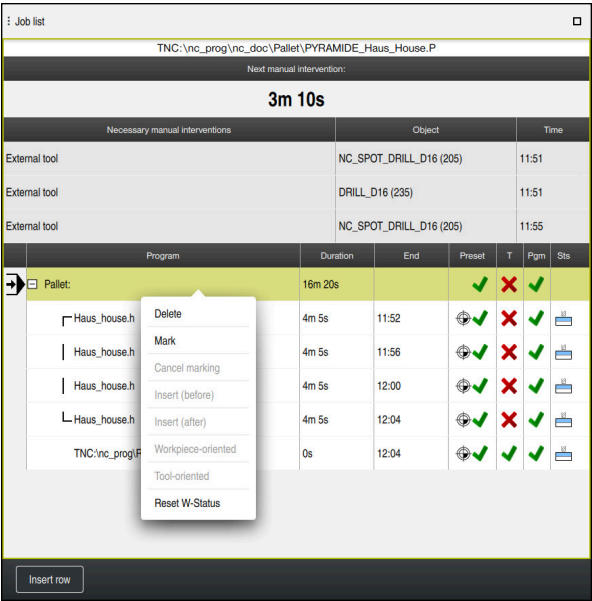
Context menu in the **Job list** workspace

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: "The Job list workspace", Page 1620

Context menu in the Program workspace
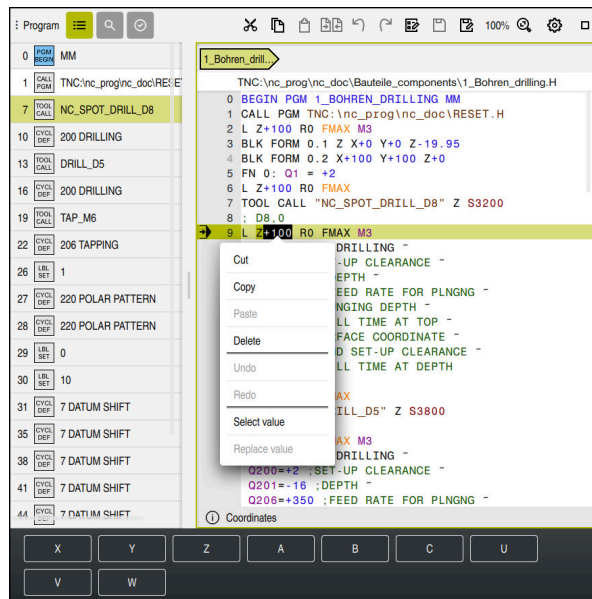
Context menu for the selected value in the **Program** workspace of the **Editor** operating mode

In the **Program** workspace, the context menu offers the following additional functions:

- **Insert last NC block**

  This function allows you to insert the most recently deleted or edited NC block. You can insert this NC block in any desired NC program.

  Only in the **Editor** operating mode and the **MDI** application

- **Create NC sequence**

  Only in the **Editor** operating mode and the **MDI** application

  Further information: "NC sequences for reuse", Page 369

- **Edit contour**

  Only in the **Editor** operating mode

  Further information: "Importing contours into graphical programming", Page 1097

- **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: "The Program workspace", Page 203

i   The **Select value** and **Replace value** functions are only available in the **Editor** operating mode and in 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 1179

Further information: "Status overview on the TNC bar", Page 155

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 1833

Context menu in the Insert NC function window

In the **Insert NC function** window, the context menu offers the following functions:

■ **Open path**

Open the NC function in the **All functions** area

■ **Edit**

Open the NC sequence in a separate tab

■ **Organize**

Open the path of the NC sequence in the **Files** operating mode

■ **Delete**

Delete the NC sequence

■ **Rename**

Rename the NC sequence

Further information: "The Insert NC function window", Page 215

28.11 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 provides arithmetic functions such as:

- Basic mathematical operations
- Basic trigonometric functions
- Square root
- Exponential calculation
- Reciprocal value
- Conversion between the mm and inch units of measure
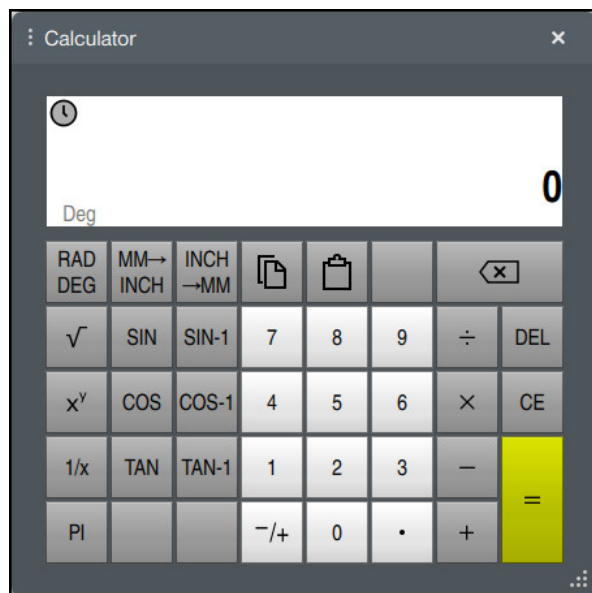
Calculator

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.

28.11.1 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.

28.11.2 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.

28.11.3 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.

28.12 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.

In OCM cycles (#167 / #1-02-1), the **OCM cutting data calculator** is available.

Further information: "OCM cutting data calculator (#167 / #1-02-1)", Page 1183

Requirement

■ Milling operation **FUNCTION MODE MILL**

Description of function

The **Cutting data calculator** window

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.

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 1182

Applying values

After the cutting data have been calculated, you can specify which values the control should apply.

You can choose among the following options for the tool:

- **Tool number**
- **Tool name**
- **Do not apply values**

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:

- **Feed per tooth (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 1724

- Tool cutting material in table **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 1724

- 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 1725
  >
  > 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 1726

- 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

28.12.1 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.

28.12.2 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

▶ Select **Apply**

> The control applies the calculated values in the NC block.

28.13 OCM cutting data calculator (#167 / #1-02-1)

28.13.1 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.

ℹ 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.

28.13.2 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.
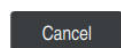
| Cancel |
or

▶ 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.

28.13.3 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

> **i** 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
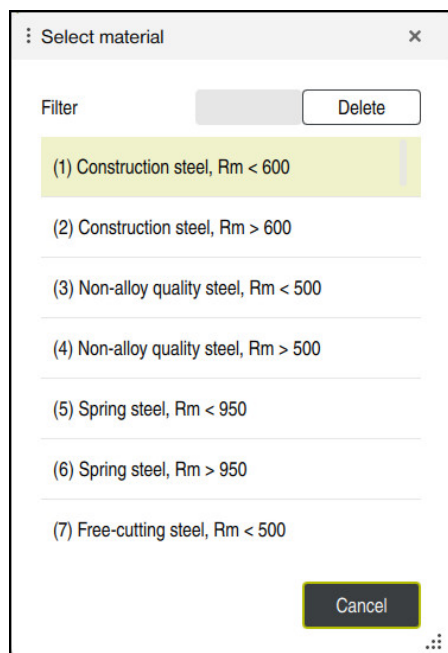
To select the workpiece material:

▶ 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.

ℹ   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

T ▲	NAME	R	DR	LCUTS	CU
1	MILL_D2_ROUGH	1	0	20	
2	MILL_D4_ROUGH	2	0	20	
3	MILL_D6_ROUGH	3	0	20	
4	MILL_D8_ROUGH	4	0	30	
5	MILL_D10_ROUGH	5	0	30	
6	MILL_D12_ROUGH	6	0	30	
7	MILL_D14_ROUGH	7	0	30	
8	MILL_D16_ROUGH	8	0	40	
9	MILL_D18_ROUGH	9	0	40	

(Left panel: all tools, tools in ..., all tool t..., milling t..., drilling t..., tapping ..., threadm..., turning t..., touchpr..., dressing..., grinding...)

You can choose the tool either by selecting it from the tool table **tool.t** or by entering the data manually.

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**

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**

i 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.t**tool table!
- 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 feed rate. 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 **i** • 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 1191

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**

> **i** 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.

28.13.4 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.

> **i**　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.

28.13.5 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	-	

28.14 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 clear 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.
⏻	Error Reset type	The control must be restarted. This message cannot be cleared.
●	Error Emergency-stop type	The control performs an emergency stop. An error message can only be cleared after the cause has been eliminated.
❗	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.
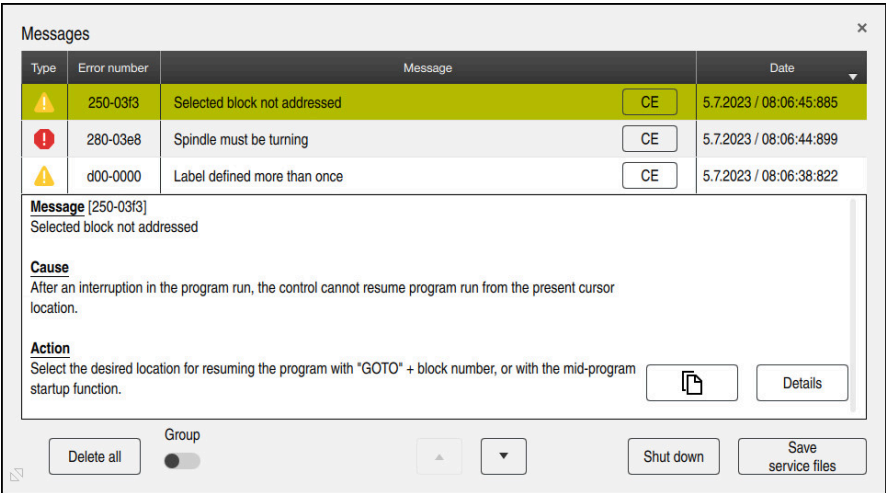
Expanded message menu with pending messages

The control displays all pending messages in chronological order.
The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (root cause, correction, information on the NC program)

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** toggle 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 clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

■ If an error occurs, you can create a service file manually.
Further information: "Creating a service file manually", Page 1195

■ If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.
Further information: "Creating a service file automatically", Page 1196

Service files help service technicians in troubleshooting the problem. 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.

The file name of each service file consists of a user-defined name and a timestamp.

If you create multiple service files with the same name, the control saves a maximum of five files and then deletes the file with the oldest timestamp, if necessary. Make a backup of the service files you created (e.g., by moving them to a different folder).

28.14.1 Creating a service file manually

To create a service file manually:

▶ Expand the message menu

▶ Select **Save service files**
> The control opens the **Save service file** window.
▶ Enter the file name

▶ Press **OK**
> The control saves the service file in the **TNC:\service** directory.

28.14.2 Creating a service file automatically

You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

To specify a new error number:

▶ Expand the message menu

Save
service files

▶ Select **Save service files**

> The control opens the **Save service file** window.

Setting for autosave

▶ Select **Setting for autosave**

> The control opens a table of error numbers.

▶ Enter the desired error number

▶ Enable the **Active** check box

> If the error occurs, the control automatically creates a service file.

▶ Enter a comment, if applicable (e.g., to describe the problem)

29

The Simulation Workspace

29.1 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 244

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 query depends on the simulation settings and the status of the active simulation.

Further information: "The Simulation settings window", Page 1204

The functions available in the simulation depend on the following settings:

- Selected type of model, for example **2.5D**
- Selected model quality, for example **Medium**
- Selected mode, for example **Machine**
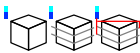
Icons in the Simulation workspace

The following icons are shown in the **Simulation** workspace:

Icon	Meaning
☰	Open or close the **Visualization options** column **Further information:** "The Visualization options column", Page 1200
🧰	Open or close the **Workpiece options** column **Further information:** "The Workpiece options column", Page 1202
🧊	Open or close the **Pre-defined views** selection menu **Further information:** "Pre-defined views", Page 1208
📝	**Save as** Export simulated workpiece as STL file **Further information:** "Exporting a simulated workpiece as STL file", Page 1209
⚙	Open or close the **Simulation settings** window **Further information:** "The Simulation settings window", Page 1204
👁	Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) DCM active
🚫	DCM inactive **Further information:** "The Visualization options column", Page 1200
⚠	DCM active with reduced minimum distance (#140 / #5-03-2) **Further information:** "Reduce the minimum clearance for DCM with FUNCTION DCM DIST (#140 / #5-03-2)", Page 878
✓++	Status of the **Advanced checks** function **Further information:** "The Visualization options column", Page 1200
🧊	Model quality **Further information:** "The Simulation settings window", Page 1204
T 0	Number or name of the current tool ℹ The display depends on the workspace size.
00:00:00	Current program run-time

The Visualization options column

In the **Visualization options** column, you can define the following display modes and functions:
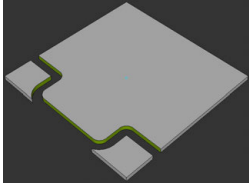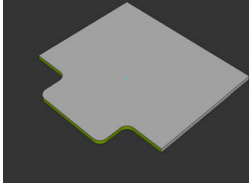
Icon or toggle switch	Meaning	Requirements
	Select the **Machine** or **Workpiece** mode	
	In the **Workpiece** mode, the control displays the workpiece, the tool, and the tool carrier. Depending on the selected mode, different functions are available, such as a display of the setup situation.	
	If you select the **Machine** mode, the control additionally displays the setup situation and the machine.	
Workpiece position	Use this function to define the position of the workpiece preset for the simulation. You can use a button to apply a workpiece preset from the preset table.	■ The **Editor** operating mode
	Further information: "Preset management", Page 705	
	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	
	You can select between the following display modes for the tool:	
	■ **Original**: Shaded, opaque representation	
	■ **Semitransparent**: Transparent representation	
	■ **Invisible**: The object is hidden	
	You can select between the following display modes for the workpiece:	
	■ **Original**: Shaded, opaque representation	
	■ **Semitransparent**: Transparent representation	
	■ **Invisible**: The object is hidden	
	You can show the tool paths during the simulation. The control displays the center-line path of the tools.	■ The **Workpiece** mode
	You can choose between the following display modes for the tool paths:	■ The **Editor** operating mode
	■ **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	
Clamping situation	Use this toggle switch to show the worktable and fixture, if required.	■ The **Workpiece** mode
DCM	Use this toggle switch to activate or deactivate Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)) for simulation.	■ The **Editor** operating mode
	Further information: "Dynamic Collision Monitoring (DCM) in the Editor operating mode", Page 852	■ Simulation reset or not started yet

Icon or toggle switch	Meaning	Requirements
Advanced checks	If the **Advanced checks** toggle switch is active, the following checks can be performed: ■ **Rapid-traverse cut** ■ **Workpiece collision** ■ **Fixture collision** **Further information:** "Advanced checks in the simulation", Page 880	■ The **Editor** operating mode
Program run options	If you activate this toggle switch, the control opens the **Program run options** window with the following selection options: ■ **Perform conditional stop** The control provides the following breakpoints: ■ **Before switch to rapid traverse** ■ **Before switch to feed rate** ■ **Between two rapid traverses** ■ **Before tool call** ■ **Before tilting the working plane** ■ **Before cycle call** ■ **In cycle call** **Further information:** "Breakpoints", Page 1761 ■ **Skip block** If an NC block is preceded by a **/** character, then the NC block is hidden. If you activate the **Skip block** toggle switch, the control skips all hidden NC blocks in the simulation. **Further information:** "Hiding NC blocks", Page 1165 If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 205 ■ **Pause at M1** If you activate this toggle switch, the control pauses the simulation at each **M1** M function in the NC program. **Further information:** "Overview of miscellaneous functions", Page 965 If the toggle switch is inactive, then the control dims the **M1** syntax element. **Further information:** "Appearance of the NC program", Page 205	■ The **Editor** operating mode

The Workpiece options column

In the **Workpiece options** column, you can define the following simulation functions for the workpiece:

Toggle switch or button	Meaning	Requirements
Measuring	Use this function to measure any points on the simulated workpiece. The control measures the distance of the measured surface to the finished part for the **3D** model type only. **Further information:** "Measuring function", Page 1211	■ The **Workpiece** mode ■ Type of model: **2,5D** or **3D**
Cutout view	Use this function to cut through the simulated workpiece along a plane. **Further information:** "Cutout view in the simulation", Page 1213	■ The **Workpiece** mode ■ The **Editor** operating mode ■ Type of model: **2,5D**
Highlight workpiece edges	Use this function to highlight the edges of the simulated workpiece.	■ The **Workpiece** mode ■ Type of model: **2.5D**
Workpiece blank frame	Use this function to show the outside lines of the workpiece blank.	■ The **Workpiece** mode ■ The **Editor** operating mode ■ Type of model: **2.5D**
Finished part	Use this function to display a finished part that was defined by means of the **BLK FORM FILE** NC function. **Further information:** "Cutout view in the simulation", Page 1213	
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:** "The Simulation settings window", Page 1204	■ The **Editor** operating mode
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**	■ Type of model: **2.5D** ■ **Model comparison** function in the **Workpiece** mode only ■ **Monitoring** function in

Toggle switch or button	Meaning	Requirements
	The control displays a comparison between the workpiece blank and the finished part. **Further information:** "Model comparison", Page 1215 The control displays a heat map on the workpiece: ■ Component heatmap with **MONITORING HEATMAP** (#155 / #5-02-1) **Further information:** "Component monitoring with MONITORING HEATMAP (#155 / #5-02-1)", Page 908 **Further information:** "Cycles for monitoring", Page 910	the **Program Run** operating mode only
Reset the workpiece	Use this function to reset the workpiece back to the workpiece blank	■ The **Editor** operating mode ■ Type of model: **2.5D**
Reset the tool paths	Use this function to reset the simulated tool paths.	■ The **Workpiece** mode ■ The **Editor** operating mode
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	■ The **Editor** operating mode ■ Type of model: **3D**

The Simulation settings window

The **Simulation settings** window is available only in the **Editor** operating mode.

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

Area	Function
General	■ **Model type** ■ **None**: quick line graphics without 3D representation ■ **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** ■ **Optimized saving of STL** (#152 / #1-04-1) If you activate the toggle switch, the control exports a simplified STL file. During this process, the control removes unnecessary triangles and simplifies the 3D model to max. 20 000 triangles. You can use the simplified STL file within **BLK FORM FILE** without any additional adaptation. **Further information:** "STL file as workpiece blank with BLK FORM FILE", Page 249 ■ **End current simulation without prompt** If the toggle switch is inactive and you open the **Simulation** workspace on a new tab, the control will show the **Close current simulation** window. You can exit the active simulation or cancel the process. If you activate the toggle switch, the control does not show the window. �📄 If you open the **Simulation** workspace on a new tab while a simulation is running, the control will always show the **Cancel running simulation** window. ■ **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 1782
Traverse ranges	■ **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

Area	Function
	each axis of the machine. For example, the machine manufacturer defines traverse ranges for large machines with two separate working spaces.
	Further information: "The Workpiece options column", Page 1202
	■ **Active traverse ranges**
	This function shows the active traverse range and the values defined for within that range.
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 1683

Action bar
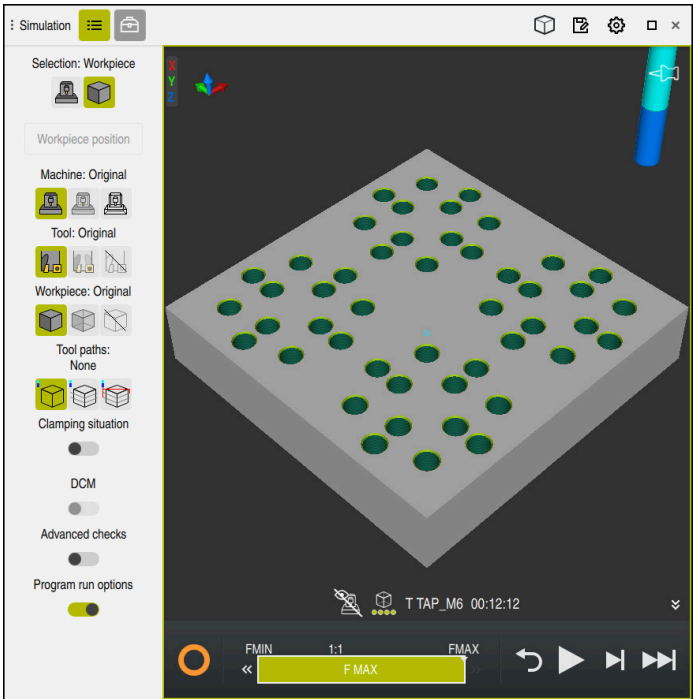
The **Simulation** workspace in the **Editor** operating mode

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.

Further information: "Display of the program run time", Page 171

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 1217
↶	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 1218

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**
- **TSHAPE**
- **R_TIP**
- 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 NC program shift the tool in the simulation.

  Further information: "Tool compensation for tool length and tool radius", Page 796

  Further information: "Tool table tool.t", Page 1684

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

29.2 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
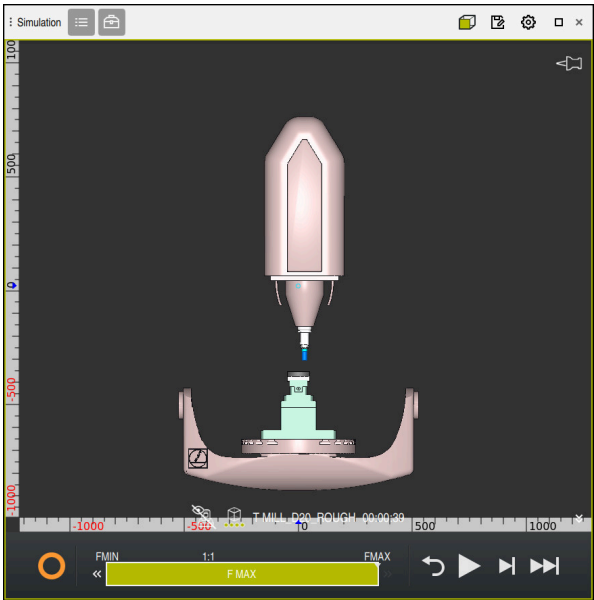
Front view of the simulated workpiece in the **Machine** mode

29.3 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 249

- Customizing STL files in **CAD Viewer** (#152 / #1-04-1)

  Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124

Description of function
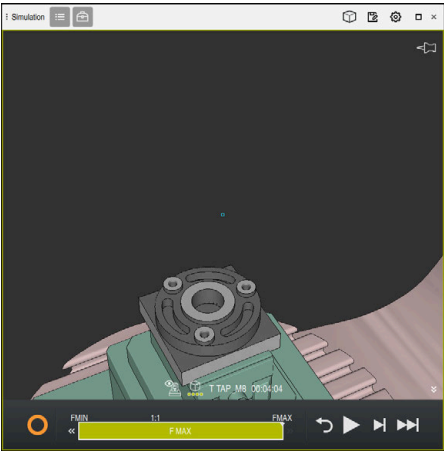
Simulated workpiece

This function can be used only in the **Editor** operating mode.

The control can only display STL files with up to 20,000 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: "The Simulation settings window", Page 1204

You can also use the **3D mesh** function to reduce the number of triangles (#152 / #1-04-1).

Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124
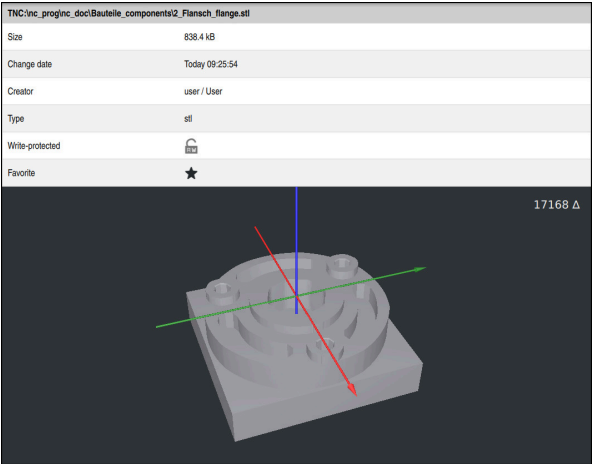
Simulated workpiece as saved STL file

29.3.1 Saving a simulated workpiece as STL file

To save a simulated workpiece as an STL file:

▶ ▶ Simulate workpiece

⚙ ▶ Select the settings as needed
  ▶ Activate **Optimized saving of STL**, if appropriate
    (#152 / #1-04-1)
  > The control simplifies the STL file when saving it.

💾 ▶ Select **Save**
  > The control opens the **Save as** window.
  ▶ Enter the desired file name
  ▶ Select **Create**
  > The control saves the created STL file.

Further information: "The Simulation settings window", Page 1204

29.4 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

- The **Workpiece** mode

Description of function

If you measure a point on the simulated workpiece, the cursor always locks onto the currently selected surface.
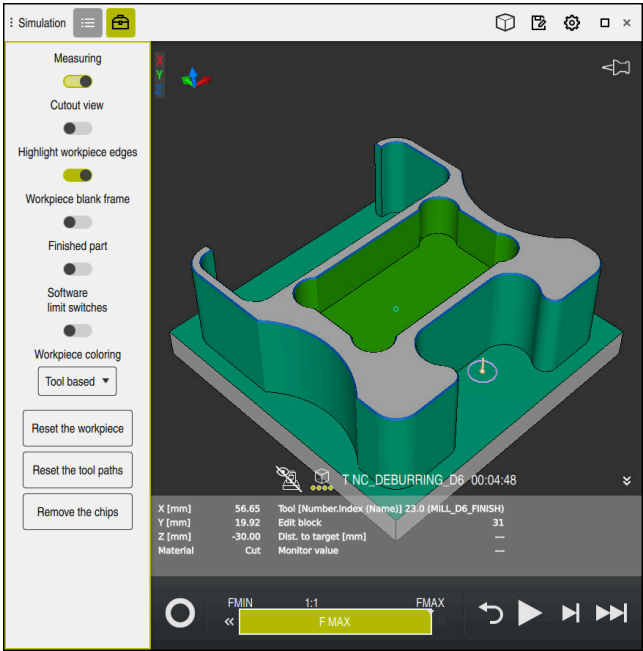
Measured point on simulated workpiece

The control shows the following information about the measured surface:

- Measured positions in the **X**, **Y** and **Z** axes, relative to the workpiece coordinate system **W-CS**
  Further information: "Workpiece coordinate system W-CS", Page 697
- 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 (#155 / #5-02-1)
  Further information: "Component monitoring with MONITORING HEATMAP (#155 / #5-02-1)", Page 908

29.4.1 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** toggle switch

▶ Select the **Workpiece coloring** selection menu

[Model comparison ▼] ▶ 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.

29.5 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

■ The **Workpiece** mode

Description of function

The Cutout view can be used in the **Editor** mode only.

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.

29.5.1 Shifting the sectional plane

To shift the sectional plane:

▶ Select the **Editor** operating mode

▶ Open the **Simulation** workspace

▶ Select the **Visualization options** column

▶ Select the **Workpiece** mode

> The control shows the workpiece view.

▶ Select the **Workpiece options** column

▶ Activate the **Cutout view** toggle 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.
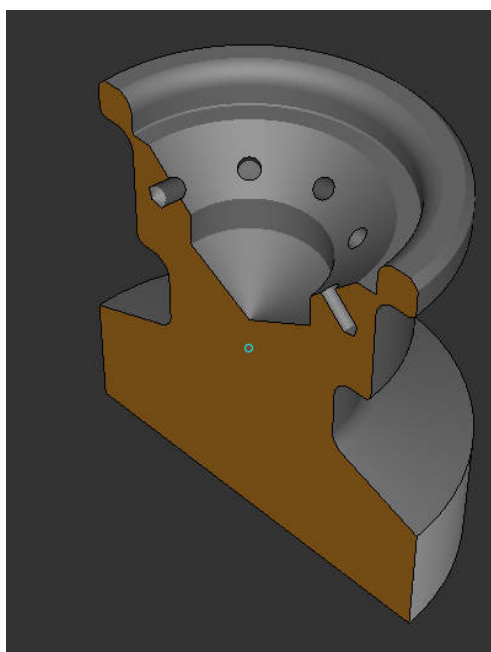
Simulated workpiece in the **Cutout view**

29.6 Model comparison

Application

With the **Model comparison** function you can compare the workpiece blank and the finished part in STL or M3D format.

Related topics

■ Programming the blank and finished part with STL files

   Further information: "STL file as workpiece blank with BLK FORM FILE", Page 249

Requirements

■ STL file or M3D file of workpiece blank and finished part
■ The **Workpiece** mode
■ Workpiece blank definition with **BLK FORM FILE**

Description of function
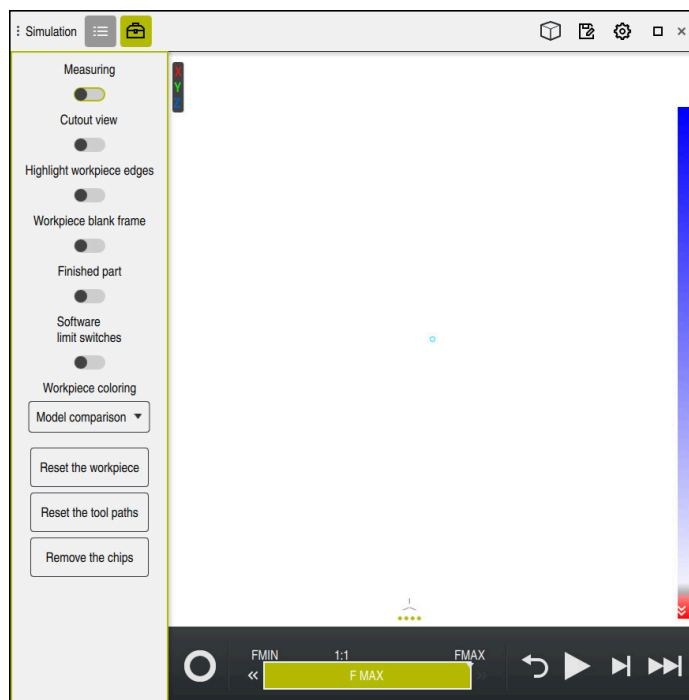
The control uses the **Model comparison** function to show the difference in material between the models being compared. The control uses a color transition from white to blue to show the difference in material. The more material there is covering the finished part model, the deeper the blue is. When 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 1213

29.7 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.

29.7.1 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".

29.8 Simulation speed

Application

You can use a slider to select any speed for the simulation.
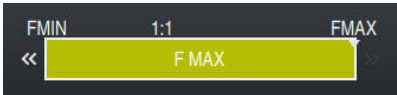
Description of function

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**)

29.9 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 1206
- Simulation speed

  Further information: "Simulation speed", Page 1217

Description of function

This function can be used only in the **Editor** operating mode.
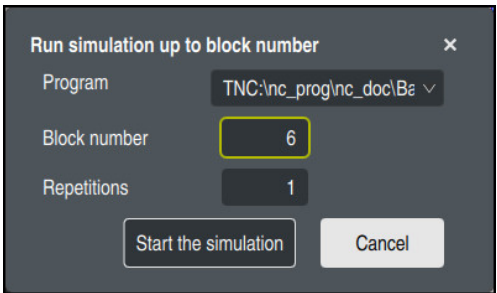
The **Run simulation up to block number** window with a defined NC block

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 363

29.9.1 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

Start the simulation

▶ Select **Start the simulation**

> The control simulates the workpiece up to the selected NC block.

30

The MDI Application

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 remembers modally effective program information.

Related topics

- Creating NC programs
  Further information: "Programming fundamentals", Page 199
- Running NC programs
  Further information: "Program Run", Page 1637

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**.
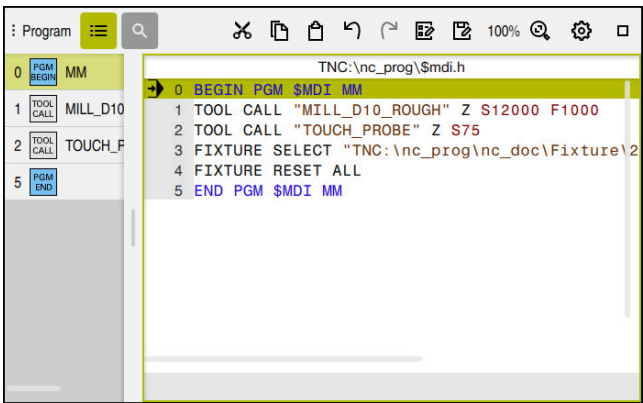
The **Program** workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **Help**
- **Positions**
  Further information: "The Positions workspace", Page 149
- **Program**
  Further information: "The Program workspace", Page 203
- **Simulation**
  Further information: "The Simulation Workspace", Page 1197
- **Status**
  Further information: "The Status workspace", Page 157
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 1160

Icons and buttons

In the **MDI** application, the function bar provides the following buttons:

Icon or button	Meaning
→	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Klartext editor	If this toggle switch is active, then you are using dialog-guided programming. If this toggle switch is not active, then you are programming in the text editor. **Further information:** "Inserting and editing NC functions", Page 217
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Areas of the Insert NC function window", Page 216
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:** "The Q parameter list window", Page 1010
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** "GOTO function", Page 1163
/ Skip block Off/On	Hide NC blocks with **/**. NC blocks hidden with a **/** character will be ignored during program run as soon as the **Skip block** toggle switch is active. **Further information:** "Hiding NC blocks", Page 1165
Skip block	If the toggle switch is active, then the control does not execute any NC blocks dimmed with the **/** character. **Further information:** "Hiding NC blocks", Page 1165 If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** "Appearance of the NC program", Page 205
; 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 1164
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 1642
F limited	Use this option to activate or deactivate the feed-rate limit for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 1774
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 893
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. **Further information:** "Retraction with stopped NC program", Page 500
Edit	The control opens the context menu. **Further information:** "Context menu", Page 1174

Icon or button	Meaning
Tools	The control opens the **Tool management** application in the **Tables** operating mode.
	Further information: "Tool management ", Page 269
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 1643
Reset program	If you select **Internal stop**, the control activates this button.
	The control resets any modally active program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always execute NC blocks in **Single Block** mode.
After the control has executed an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 1643

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 **Q parameter list** window
  - 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 of an NC program with **PGM CALL**
  - Test run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** functions while program run is interrupted
  - **Block scan** function
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

31

Touch Probes

31.1 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 (#17 / #1-05-1)", Page 1694

- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)

  Further information: "Machine parameters", Page 1833

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: "The Settings Application", Page 1777

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.
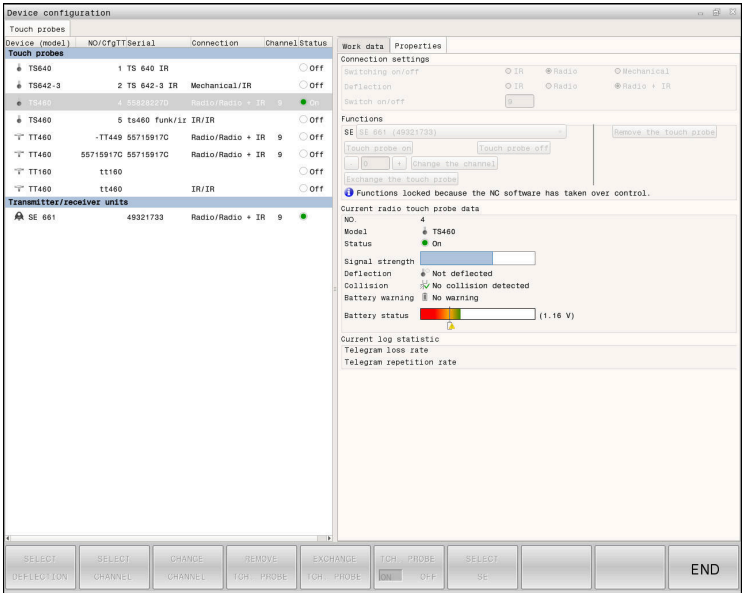
Areas of the **Device configuration** window

The **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.

The 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.

The 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
Type	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

> **i** 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 DEFLEC-TION	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 IR POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT RADIO 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.

31.2 Calibrating a workpiece touch probe (#17 / #1-05-1)

31.2.1 Overview

The control provides calibration cycles for calibrating the length and the radius:

Cycle		Call	Further information
460	**CALIBRATION OF TS ON A SPHERE** (#17 / #1-05-1) ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	**DEF**-active	Page 1234
461	**TS CALIBRATION OF TOOL LENGTH** (#17 / #1-05-1) ■ Calibrating the length	**DEF**-active	Page 1242
462	**CALIBRATION OF A TS IN A RING** (#17 / #1-05-1) ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	**DEF**-active	Page 1244
463	**TS CALIBRATION ON STUD** (#17 / #1-05-1) ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	**DEF**-active	Page 1247

31.2.2 Fundamentals

Application

> ⚙ The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
>
> HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

In order to precisely specify the actual trigger point of a 3D 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.

Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3D 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 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.

> ⓘ - The control applies 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 (#17 / #1-05-1)", Page 1694

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.

31.2.3 Cycle 460 CALIBRATION OF TS ON A SPHERE (#17 / #1-05-1)

ISO programming
G460

Application

> ⚙ Refer to your machine manual.
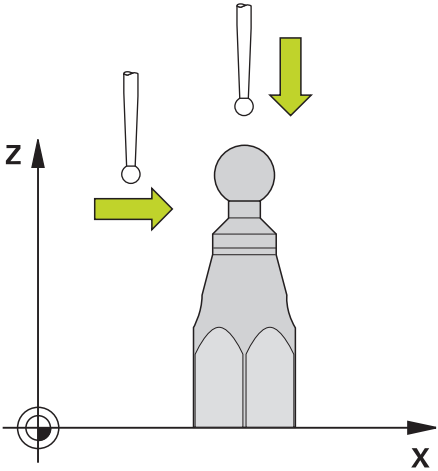
Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

With Cycle **460** you can calibrate a triggering 3D touch probe automatically using an exact calibration sphere.

Before calibrating with a normal stylus:

Before starting the calibration cycle, you need to pre-position the touch probe:

▶ Define the approximate value of the radius R and length L of the touch probe

▶ In the working plane, center the touch probe above the calibration sphere

▶ Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance above the calibration sphere. The set-up clearance consists of the value from the touch probe table plus the value from the cycle.
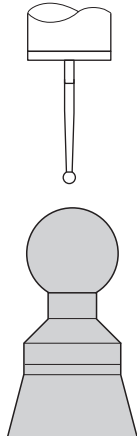
Pre-positioning with a normal stylus

Before calibrating with an L-shaped stylus:

▶ Clamp the calibration sphere

> ⓘ It must be possible to probe the north pole and south pole during calibration. If this is not possible, the control cannot determine the sphere radius. Ensure that no collision can occur.

▶ Define the approximate value of the radius **R** and length **L** of the touch probe. You can determine these with a tool presetter.

▶ Enter the approximate center offset in the touch probe table:
   - **CAL_OF1**: length of the extension
   - **CAL_OF2**: **0**

▶ Load the touch probe and orient it parallel to the main axis, for example with Cycle **13 ORIENTATION**

▶ Enter the calibration angle in the **CAL_ANG** column of the tool table.

▶ Position the center of the touch probe over the center of the calibration sphere

> Since the stylus is angled, the touch probe sphere is not centered over the calibration sphere.

▶ Position the touch probe in the tool axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere
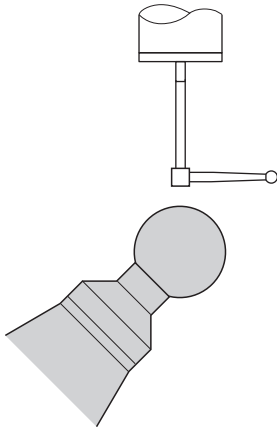
Pre-positioning with an L-shaped stylus
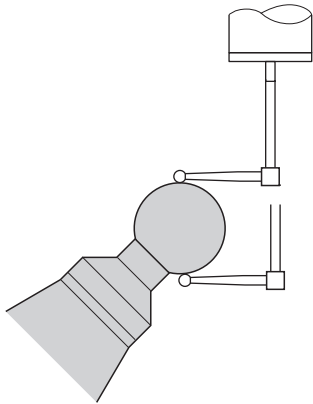
Calibration process with an L-shaped stylus

Cycle run
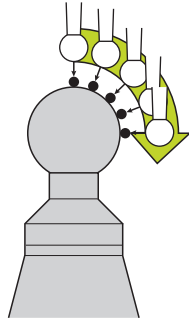
The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

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 controls positions the touch probe in the 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 determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
7 Once **CAL_ANG** has been determined, the radius calibration begins
8 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 determination of the spindle angle for calibration **CAL_ANG** begins (for L-shaped stylus)
7 Once **CAL_ANG** has been determined, the radius calibration begins
8 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned
9 The control determines the length of the touch probe at the north pole of the calibration sphere
10 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

i
- For an L-shaped stylus, the calibration takes place between the north and south pole.
- 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 performing length calibration with Cycle **460** is not recommended!
- One application example for calibrating the length with Cycle **460** is the comparison of two touch probes

Notes

> ⚙ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
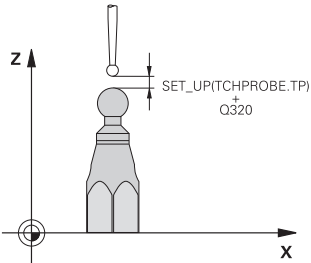
▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.

▶ 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, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- In order to achieve optimum accuracy results with an L-shaped stylus, HEIDENHAIN recommends calibrating and probing at identical speeds. Note the setting of the feed override if it is active for probing.
- If you program **Q455=0**, the control will not perform a 3D calibration.
- If you program **Q455=1** to **30**, the control will perform a 3D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles.
- 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

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 active 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)? Define how the touch probe will move between the measuring points: **0**: Move to measuring height between measuring points **1**: Move to clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. This 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. This 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. This 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 3D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3D calibration. During 3D 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 3D 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.

31.2.4 Cycle 461 TS CALIBRATION OF TOOL LENGTH (#17 / #1-05-1)

ISO programming

G461

Application

⚙ Refer to your machine manual.
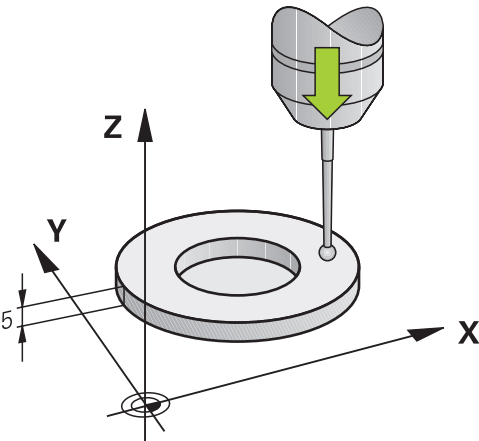
Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above the calibration ring.

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 guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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
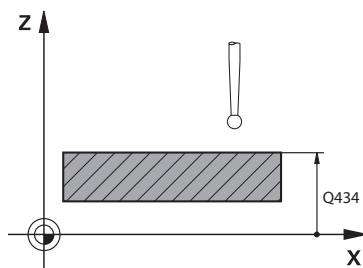
Help graphic	Parameter
	Q434 Preset for length? Preset for the length (e.g., height of the calibration ring). This value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~
Q434=+5   ;PRESET

31.2.5 Cycle 462 CALIBRATION OF A TS IN A RING (#17 / #1-05-1)

ISO programming

G462

Application

> ⚙ Refer to your machine manual.
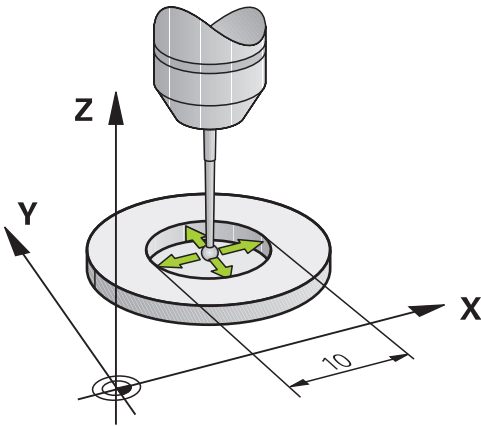
Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

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 a reversal measurement, 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 reversal measurement determines not only the radius but ralso the center offset (**CAL_OF** in the touch-probe table).

- 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 manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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 active 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. This 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. This 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

31.2.6 Cycle 463 TS CALIBRATION ON STUD (#17 / #1-05-1)

ISO programming
G463

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 stud (approximate measurement) and positions the touch probe in the center. Then, during the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe permits a reversal measurement, 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 reversal measurement determines now only the radius but also the center offset (CAL_OF in the touch-probe table).

- 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.

Whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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
SET_UP(TCHPROBE.TP) + Q320	**Q407 Radius of calibr. stud?** Diameter of the calibration stud Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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)? Define how the touch probe will move between the measuring points: **0**: Move to measuring height between measuring points **1**: Move to clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. This 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. This 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

31.3 Calibrating a tool touch probe (#17 / #1-05-1)

31.3.1 Overview

Cycle		Call	Further information
480	**CALIBRATE TT** (#17 / #1-05-1) ▪ Calibrating the tool touch probe	DEF-active	Page 1250
484	**CALIBRATE IR TT** (#17 / #1-05-1) ▪ Calibrating the tool touch probe (e.g., infrared tool touch probe)	DEF-active	Page 1253

31.3.2 Fundamentals

Application

The following cycles can be used to calibrate the tool touch probe or the infrared tool touch probe.

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 167

> **i** 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.

31.3.3 Cycle 480 CALIBRATE TT (#17 / #1-05-1)

ISO programming

G480

Application

> **⚙** Refer to your machine manual!

You calibrate the TT with touch probe cycle **480**. The calibration process runs automatically. The control also measures the center offset 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 **480**.

Cycle run

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 tooth 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

11 TOOL CALL 12 Z	
12 TCH PROBE 480 CALIBRATE TT ~	
Q260=+100	;CLEARANCE HEIGHT

31.3.4 Cycle 484 CALIBRATE IR TT (#17 / #1-05-1)

ISO programming

G484

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 run

> ⚙ 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.

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.

> ℹ Ensure that the calibration tool is located above the measuring surface of the probe contact.

- ▶ Press **NC Start** to resume cycle run
- > 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!

If you program **Q536=1**, the tool must be pre-positioned before calling the cycle. 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. There is a danger of collision!

▶ 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 tooth 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 pre-position the tool. The calibration tool approaches the position automatically.
	2: Current position of the calibration tool. See **Q523 = 0**. **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

32

Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)

32.1 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 for the workpiece
  Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
- Preset table
  Further information: "Preset table *.pr", Page 1710
- Datum table
  Further information: "Datum table *.d", Page 1721
- Reference systems
  Further information: "Reference systems", Page 690
- Preassigned variables
  Further information: "Preassigned Q parameters", Page 1013

Requirements

- Software option Touch-probe functions (#17 / #1-05-1)

  > **i** If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 1272

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**
- **Set up fixtures** (#140 / #5-03-2)

  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859
- **Set up the workpiece** (#159 / #1-07-1)

  Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 1277

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 1266
- 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 1268
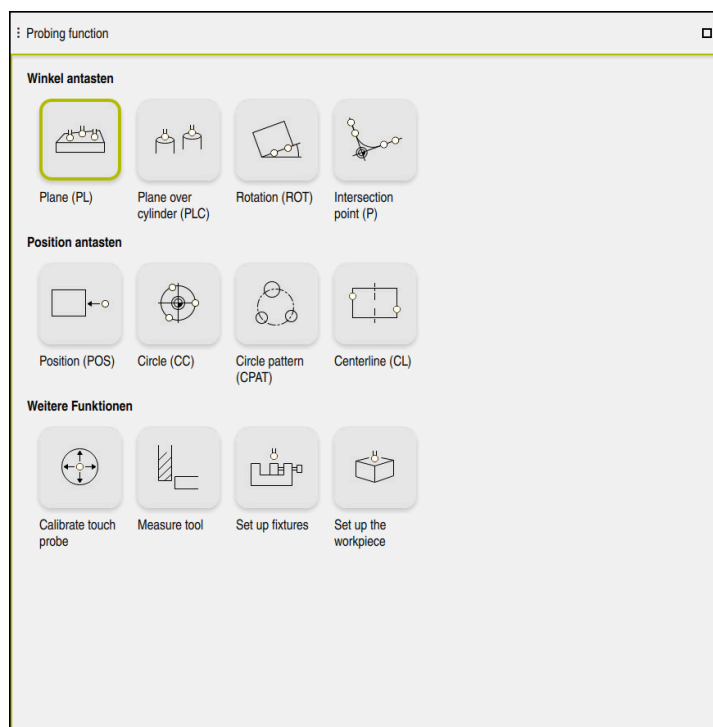
The **Probing function** workspace

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 1270
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 *.pr", Page 1710
>
> 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 1288

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 1266
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 1268
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.

The 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 1272
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.
	Further information: "Werkzeug vermessen mit Ankratzen", Page
Set up fixtures	The **Set up fixtures** function is used to determine the position of a clamping device in the working space using a workpiece touch probe (#140 / #5-03-2).
	Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859
Set up the workpiece	The **Set up the workpiece** function is used to determine the position of a workpiece in the working space using a workpiece touch probe (#159 / #1-07-1).
	Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 1277

Icons and buttons

General icons and buttons in the touch probe functions

The following buttons are available, depending on the selected touch probe function:

Icon or button	Meaning
←	**Exit probing**
⊕	Select the workpiece preset and the pallet preset and edit the values if required **Further information:** "The Change the preset window", Page 1265 **Further information:** "Preset table *.pr", Page 1710
ⓘ	Display help graphics for the selected touch probe function
Y+	Select the probing direction
⊹	Apply the actual position
○→⫿	Manually approach and probe points on a straight surface
⊕ M	Manually approach and probe points on a stud or in a hole
⊕ A	Automatically approach and probe points on a stud or in a hole 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.
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 269
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 1643

Symbols and buttons for calibration

The control offers the following functions for calibrating a 3D touch probe:

Icon or button	Meaning
⊥↓	Calibrating the length of a 3D touch probe
⊕⊗	Calibrating the radius of a 3D touch probe
Apply calibra-tion data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 1272

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Icon	Meaning
⊐⊏	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

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	Meaning
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:** "The 3-D rotation window (#8 / #1-01-1)", Page 781
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 provides the following functions:

Button	Meaning
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 *.pr", Page 1710
Correct the datum	The **Correct the datum** function transfers the measuring result into a desired line of the datum table. **Further information:** "Datum table *.d", Page 1721
Align rotary table	The **Align rotary table** function aligns the rotary axes mechanically according to the measuring result.
Correct the pallet reference point	The **Correct the pallet reference point** function transfers the measuring result into the active line of the pallet preset table. **Further information:** "Pallet preset table", Page 1635

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. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

---

The Change the preset window

In the **Change the preset** window you can select a preset or edit the values of a preset.

Further information: "Preset management", Page 705

The **Change the preset** window provides the following buttons:

Icon or button	Meaning
⊕	The control shows the preset table. **Further information:** "Preset management", Page 705
⊕ (pallet)	The control shows the pallet preset table. **Further information:** "Pallet preset table", Page 1635
Reset basic rotation	The control resets the values from the columns **SPA**, **SPB** and **SPC**.
Reset offsets	The control resets the values from the columns **A_OFFS**, **B_OFFS** and **C_OFFS**.
Apply changes and delete existing probe objects	The control activates the selected preset and rejects the touch points used so far. Then the control closes the window.
Apply	The control saves the changes and the selected preset. Then the control closes the window.
Reset	The control cancels the changes and restores the initial condition.
Cancel	The control closes the window without saving.

ℹ️ If you change a value, the control marks this value with a blue dot.

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. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

▶ Refer to the machine manufacturer's documentation
▶ Use pallet presets only in conjunction with pallets
▶ Change pallet presets only after discussion with the machine manufacturer
▶ Check the pallet preset in the **Setup** application before you start machining

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.

32.1.1 Setting a preset in a linear axis

To probe the preset in any axis:

▶ Select the **Manual** operating mode
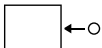
▶ Call the workpiece touch probe as a tool

▶ 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.

▶ Use the axis keys to 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**)

Compensate the active preset

▶ Select **Compensate the active preset**

> The control enters the defined nominal value in the preset table.

> The control marks the row with an icon.

> 🛈 If you use the **Correct the datum** function, the control also marks this row with an icon.
>
> 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.
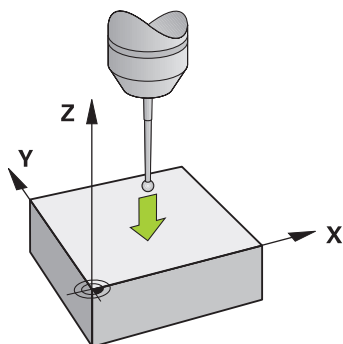
32.1.2 Determining the circle center point of a stud using the automatic probing method

To probe a circle center point:

▶ Select the **Manual** operating mode

▶ Call the workpiece touch probe as a tool
  Further information: "The Manual operation application", Page 186

▶ 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 **Safety clearance (min. value = SET_UP)** if required

> ⓘ The control suggests the total of the value in the **SET_UP** column of the touch probe table and the ball tip radius as a safety distance.

▶ 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**)

Compensate the active preset

▶ Select **Compensate the active preset**

> The control sets the preset to the entered nominal value.

> The control marks the row with an icon.

> ⓘ If you use the **Correct the datum** function, the control also marks this row with an icon.
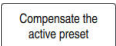
▶ Select **Exit probing**

> The control closes the **Circle (CC)** probing function.
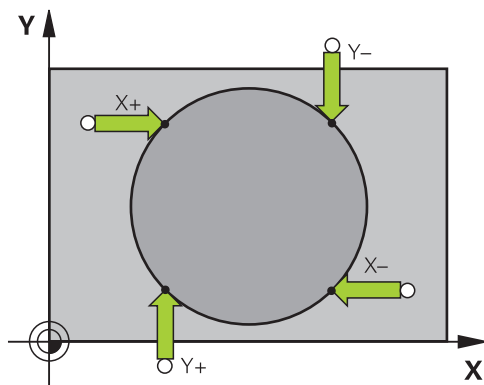
32.1.3 Determining and compensating the rotation of a workpiece

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.
> The control marks the row with an icon.

> 🛈 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.
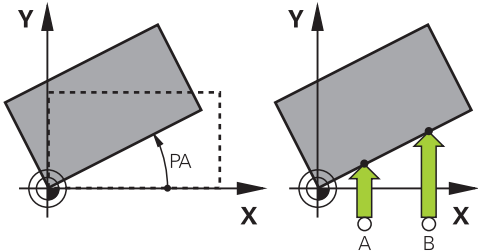
32.1.4 Using touch probe functions with mechanical probes or dial gages

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

▶ Insert the tool, such as an 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 control marks the row with an icon.
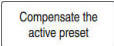
> 🛈 The determined angles have different effects depending on whether they are transferred as an offset or as a basic rotation to the corresponding table.
>
> **Further information:** "Comparison of offset and 3D basic rotation", Page 1288

▶ Select **Exit probing**
> The control closes the **Rotation (ROT)** probing function.

Notes

- If you use a non-contacting tool touch probe (such as a laser touch probe), then you are using touch-probe functions from a third-party supplier. 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.
- 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
- If the touch point is not reached during the touching process, the control will display a warning. The probing process can be continued with **NC Start**.

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.

32.2 Calibrating the workpiece touch probe

Application

The touch probe must be calibrated in order to enable determining precisely the actual switching point of a 3D touch probe. Otherwise, the control cannot ascertain exact measuring results.

Related topics

- Calibrate the workpiece touch probe automatically

  Further information: "Calibrating a workpiece touch probe (#17 / #1-05-1)", Page 1231

- Touch probe table

  Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

Description of function
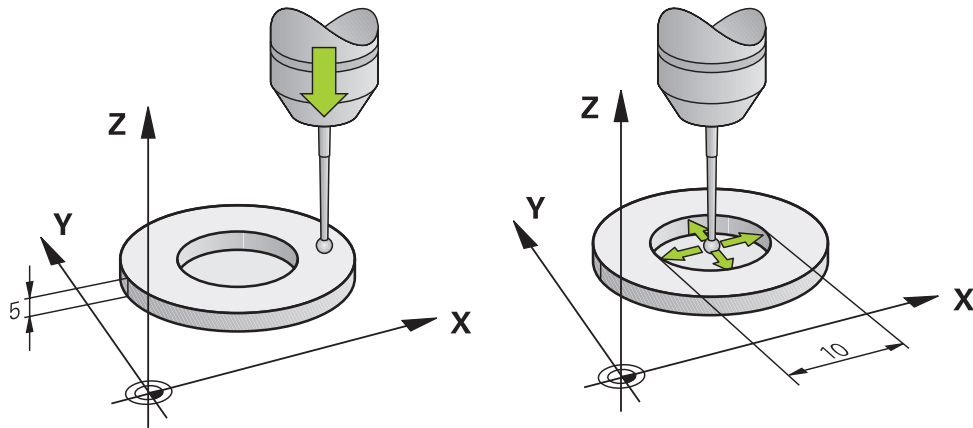
During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 253

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.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

At first, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension
	The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis
	For this, manually position the extension in the direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

When calibrating the length, the control aligns the touch probe with the calibration angle defined in the **CAL_ANG** column.

While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

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 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 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.

32.2.1 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 706

▶ Set preset in the tool axis (e.g., with **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., with **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., with **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

▶ 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.

32.2.2 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

Apply calibration data ▶ 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.

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 guarantees the proper operation of the touch probe cycles only 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 (#17 / #1-05-1)", Page 1694

32.3 Setting up the workpiece with graphical support (#159 / #1-07-1)

Application

Use the **Set up the workpiece** function to determine the position and misalignment of a workpiece with only one touch-probe function and save it as a workpiece preset. During setup, you can probe curved surfaces.

The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model.

Related topics

- Touch-probe functions in the **Setup** application
  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257
- Generating an STL file of a workpiece
  Further information: "Exporting a simulated workpiece as STL file", Page 1209
- **Simulation** workspace
  Further information: "The Simulation Workspace", Page 1197
- Setting fixtures with graphical support (#140 / #5-03-2)
  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 859

Requirements

- Software option Touch-probe functions (#17 / #1-05-1)

  > **ⓘ** If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.

- Graphically supported setup software option (#159 / #1-07-1)
- Touch probe properly defined in the tool management:
  - Spherical radius in the **R2** column
  Further information: "Tool data for touch probes (#17 / #1-05-1)", Page 267
- Workpiece touch probe calibrated
  Further information: "Calibrating the workpiece touch probe", Page 1272
- 3D model of the workpiece as STL file
  The STL file may contain up to 300,000 triangles. The more the 3D model corresponds to the actual workpiece, the higher the possible workpiece setup accuracy.
  If required, optimize the 3D model with the **3D mesh** (#152 / #1-04-1) function.
  Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 1124

Description of function

The **Set up the workpiece** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

The scope of the **Set up the workpiece** function depends on the Extended Functions Group 1 (#8 / #1-01-1) and Extended Functions Group 2 (#9 / #4-01-1) software options as follows:

■ Both software options enabled:

You can tilt before setting up and incline the tool while setting up in order to probe even complex workpieces (e.g., shaped parts).

■ Only Extended Functions Group 1 (#8 / #1-01-1) enabled:

You can tilt before setting up. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.

> **ℹ** If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.

■ None of the two software options is enabled:

You cannot tilt before setting up. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736

Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1) | Setting up the workpiece with graphical support (#159 / #1-07-1)

32

Extension of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphical support for setting up the workpiece.
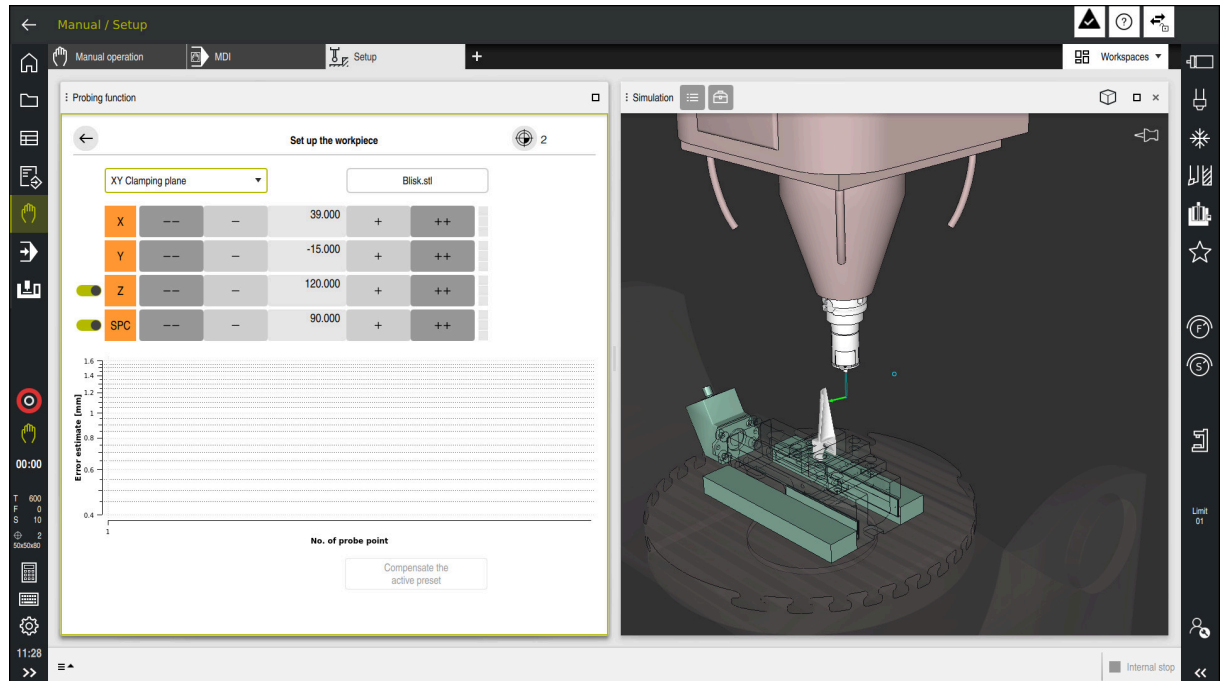
The **Set up the workpiece** function with the **Simulation** workspace open

When the **Set up the workpiece** function is active, the **Simulation** workspace displays the content below:

- Current position of workpiece as viewed by the control
- Probed points on the workpiece
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the workpiece or the workpiece touch probe is positioned within the workpiece, as seen by the control.

    In this case you can correct the position of the 3D model in the simulation, if required.
  - Red arrow

    Probing in the direction of the arrow is not possible.
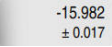
    > **i** Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.

  - Yellow arrow

    Probing in the direction of the arrow is possible to a limited extent. The probing is performed in a deselected direction or could cause collisions.
  - Green arrow

    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up the workpiece** function contains the following icons and buttons:

Icon or button	Meaning
⊕	Open the **Change the preset** window You can select the workpiece preset and the pallet preset and edit values if required. ℹ️ After the first point has been probed, the control dims the icon.
XY Clamping plane	Use this selection menu to define the probing mode. Depending on the probing mode, the control displays the respective axis directions and spatial angles. **Further information:** "Probing mode", Page 1281
1_plate_blk.stl	File name of 3D model
– –	Shifts the position of the virtual workpiece by 10 mm or 10° in the negative axis direction ℹ️ Shifts the workpiece in mm in a linear axis and in degrees in a rotary axis.
–	Shifts the position of the virtual workpiece by 1 mm or 1° in the negative axis direction
-15.982 ± 0.017	■ Enter the position of the virtual workpiece directly ■ Value and estimated accuracy of the value after the probing
+	Shifts the position of the virtual workpiece by 1 mm or 1° in the positive axis direction
+ +	Shifts the position of the virtual workpiece by 10 mm or 10° in the positive axis direction
	Status of direction The control displays the following colors: ■ Gray The axis direction is deselected in this setup process and is not considered. ■ White No touch points have been determined yet. ■ Red The control cannot locate the workpiece position in this axis direction. ■ Yellow The position of the workpiece in this axis already contains information. The information is not meaningful yet. ■ Green The control can locate the workpiece position in this axis direction.
Compensate the active preset	The control saves the determined values in the active row of the preset table.

Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1) | Setting up the workpiece with graphical support (#159 / #1-07-1)

32

Probing mode

The following modes for probing the workpiece are available to you:

- **XY Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPC**

- **XZ Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPB**

- **YZ Clamping plane**

  X, **Y** and **Z** axis directions as well as spatial angle **SPA**

- **6D**

  X, **Y** and **Z** axis directions as well as spatial angles **SPA**, **SPB** and **SPC**

Depending on the probing mode, the control displays the respective axis directions and spatial angles. In the **XY**, **XZ** and **YZ** clamping planes a toggle switch allows you to deselect the respective tool axis and spatial angle, if required. The control will not take deselected axis directions into account in the setup process and positions the workpiece by considering the remaining axis directions only.

HEIDENHAIN recommends executing the setup process as follows:

1 Pre-position a 3D model in the machine's working space

   At this point in time, the control does not know the precise position of the workpiece, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe produces values close to the position of the real workpiece.

2 Set the first touch points in the **X**, **Y** and **Z** axis directions

   If the control can determine the position in one axis direction, it will change the status of that axis to green.

3 Determine the spatial angle by setting further touch points

   To achieve maximum accuracy when probing the spatial angles, the touch points should be as far apart from one another as possible.

4 Increase the accuracies by additional check points

   Additional check points at the end of the measuring process improve the matching accuracy and minimize the misalignment between the 3D model and the real workpiece. Perform as many probing processes as necessary until the control displays the desired accuracy beneath the current value.

The error estimate diagram shows for each touch point the approximate distance of the 3D model from the real workpiece.

Further information: "Error estimate diagram", Page 1282
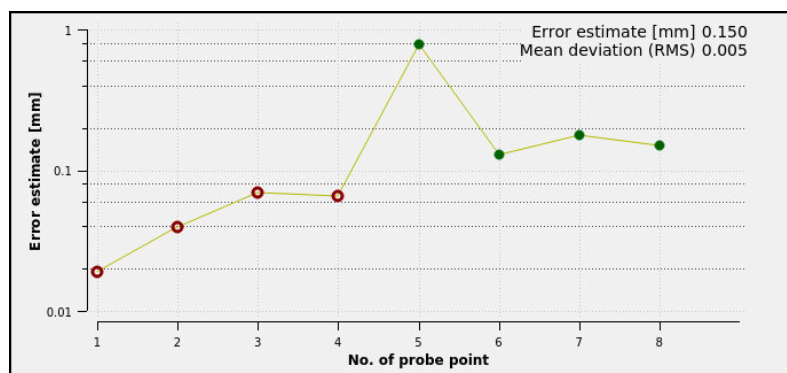
Error estimate diagram

Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated value of the distance of the 3D model from the real workpiece. For this purpose, the control considers not only the touch points, but the whole workpiece.

When the error estimate diagram shows green circles and the desired accuracy, the setup process will be complete.

The following factors influence the accuracy that can be achieved when measuring workpieces:

- Accuracy of workpiece touch probe
- Accuracy of the machine kinematic configuration
- Deviations of the 3D model from the real workpiece
- Condition of the actual workpiece (e.g., unmachined areas)

Error estimate diagram in the **Set up the workpiece** function

The error estimate diagram of the **Set up the workpiece** function shows the following information:

- **Mean deviation (RMS)**

  This area shows the average distance of the real workpiece from the 3D model in mm.

- **Error estimate [mm]**

  This axis shows the course of the error estimate based on the individual touch points. The control shows red circles until it can determine all axis directions. From then on the control will show green circles.

- **No. of probe point**

  This axis shows the numbers of the individual probing points.

Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1) | Setting up the workpiece with graphical support (#159 / #1-07-1)

32

32.3.1 Setting up the workpiece
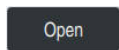
Use the **Set up the workpiece** function to set the preset:

▶ Affix a real workpiece in the machine's working space

▶ Select the **Manual** operating mode

▶ Insert the workpiece touch probe

▶ Manually position the workpiece touch probe above the workpiece at a notable point (e.g., a corner)

> **ℹ** This step makes the subsequent steps easier.

▶ Select the **Setup** application

▶ Select **Set up the workpiece**

> The control opens the **Set up the workpiece** menu.

▶ Select a 3D model matching the real workpiece

▶ Select **Open**

> The control opens the selected 3D model in the simulation.

▶ If necessary, open the **Change the preset** window

▶ Select a new preset if necessary

▶ Select the **Apply** function if necessary

▶ Pre-position the 3D model by using the buttons for the individual axis directions within the virtual working space of the machine

> **ℹ** For pre-positioning the workpiece, use the workpiece touch probe as a point of reference.
>
> Even during the setup process, the shift functions are available for correcting the fixture position manually. Then, probe a new point.

▶ Specify the probing mode (e.g., **XY Clamping plane)**

▶ Position the workpiece touch probe until the control shows a green arrow pointing downward

> **ℹ** 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 workpiece will actually be probed. Check if the workpiece position in the simulation and in the machine match each other and if probing in the direction of the arrow is possible on the machine.
>
> Do not probe 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 workpiece to the probed position. The control marks the probed position with a point in the simulation.

▶ Repeat this process in axis directions **X+** and **Y+**

> The control changes the status of the axes to green.

▶ Probe another point in axis direction **Y+** for the basic rotation

> The control changes the status of the **SPC** spatial angle to green.

▶ Probe the check point in axis direction **X-**

Compensate the active preset

▶ Select **Compensate the active preset**

> The control saves the determined values in the active row of the preset table.

← ▶ Exit the **Set up the workpiece** function

Notes

NOTICE

Danger of collision!

To probe the clamping situation in the machine exactly, 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 workpiece.
- Collisions might be more easily detected, if a tool carrier is assigned to the workpiece touch probe.
- HEIDENHAIN recommends probing check points for one axis direction on both sides of the workpiece. As a result, the control will correct the position of the 3D model in the simulation uniformly.

32.4 Measuring the tool by scratching

Application

Not all machines are equipped with a tool touch probe for measuring a tool. The **Tool measured** touch probe function enables determining the tool dimensions by scratching a workpiece.

Related topics

- Touch probe functions in the **Setup** application

  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257

- Measuring the tool automatically with cycles

  Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555

Requirement

- Software option Touch-probe functions (#17 / #1-05-1)

> **i** If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (#17 / #1-05-1) is automatically enabled.

Description of function

You do not use a 3D touch probe for scratching, but the tool to be measured. In the scratching process, approach the tool carefully to a workpiece surface until you can see a thin chip being removed. The handwheel allows obtaining a higher accuracy.

In the **X** or **Y** probing directions, the tool radius can be determined. When selecting probing direction **Z**, the tool length is measured.

Buttons in the Measure the tool function

The control offers the following options for writing the measured radius or length values into the tool table:

Button	Meaning
Write basic values	The control transfers the values into columns **R** or **L**. The control resets existing delta values in columns **DR** or **DL**.
Write delta values	The control enters the delta values in columns **DR** or **DL**.

**Further information:** "Tool tables", Page 1683

32.4.1 Tool measurement by scratching

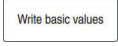The dimensions of an end mill can be determined by using the **Tool measured** function as follows:

▶ Select the **Manual** operating mode

▶ Set the workpiece preset if required

> ℹ️ Position the workpiece preset on the surfaces to be scratched in order to obtain a clear reference.

▶ Insert the tool to be measured
▶ Define the speed if required
▶ Start the tool spindle
▶ Select the **Setup** application
▶ Select the **Measure tool** probing function

▶ Scratch the workpiece in the desired axis direction (e.g., **X+**)

▶ Select the associated probing direction **X+**

▶ Select **actual position capture**
> The control transfers the actual X axis position into the **Actual value** column.
> The control shows the measurement results.
▶ Enter a **Nominal value** (e.g., **0**)
▶ Select **Write basic values**
> The control transfers the value into column **R** of the tool table.
> The control resets the existing delta value in the **DR** column.

> ℹ️ When selecting **Write delta values**, the control will enter only one delta value in column **DR**.

▶ Scratch another axis direction if required (e.g., **Z-**)

▶ Select **Exit probing**
> The control closes the **Measure tool** probing function.

32.5 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.

32.5.1 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
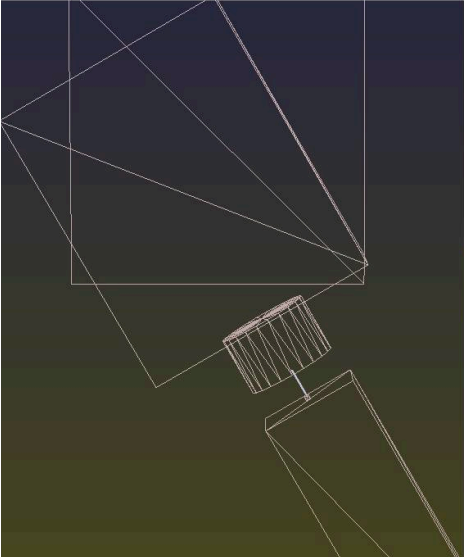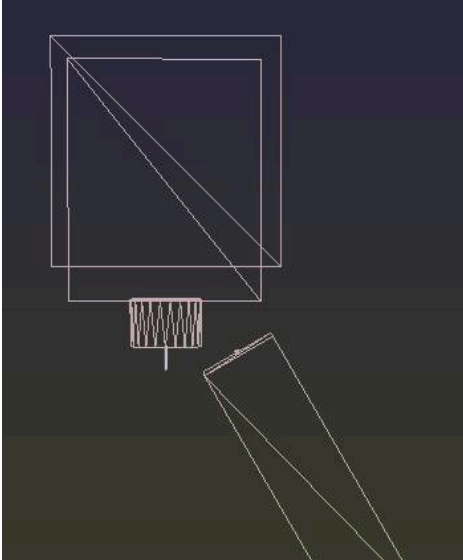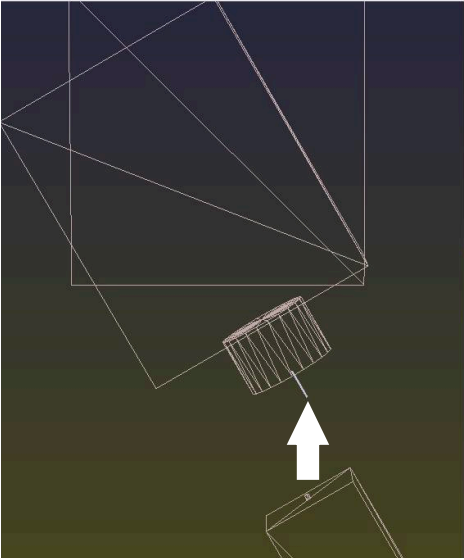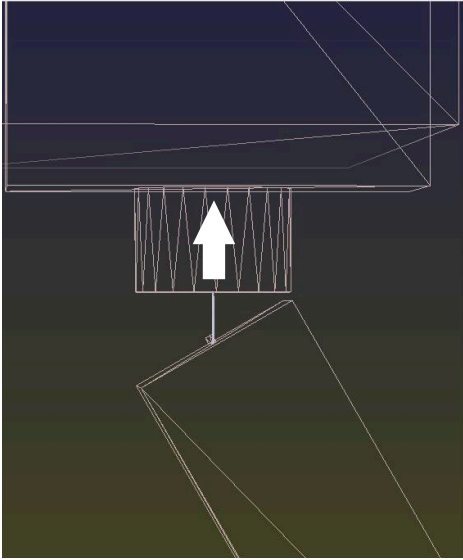
| **NOTICE** |
|---|
| **Danger of collision!** |
| While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse! |
| ▶ Carefully 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.

32.6 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

Offset	3D basic rotation
Initial state	Initial state

Position display:	Position display:
■ Actual position	■ Actual position
■ **B** = 0	■ **B** = 0
■ **C** = 0	■ **C** = 0
Preset table:	Preset table:
■ **SPB** = 0	■ **SPB** = -30
■ **B_OFFS** = -30	■ **B_OFFS** = +0
■ **C_OFFS** = +0	■ **C_OFFS** = +0

Movement in +Z without tilting	Movement in +Z without tilting

Movement in +Z with tilting	Movement in +Z with tilting
PLANE SPATIAL with **SPA+0 SPB+0 SPC +0**	**PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**
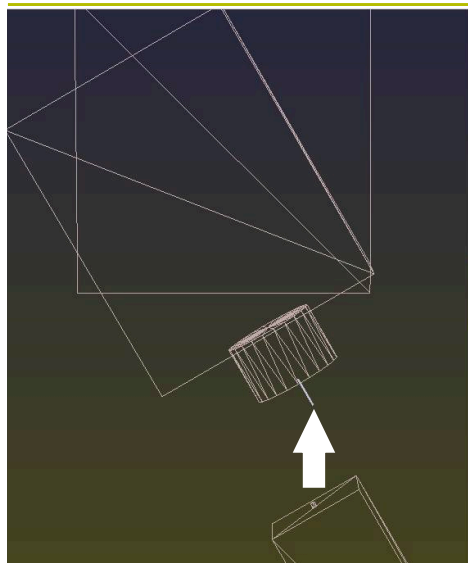
Offset	3D basic rotation
> The orientation **is not correct**!	> The orientation is correct! > The next machining step will be **correct**.
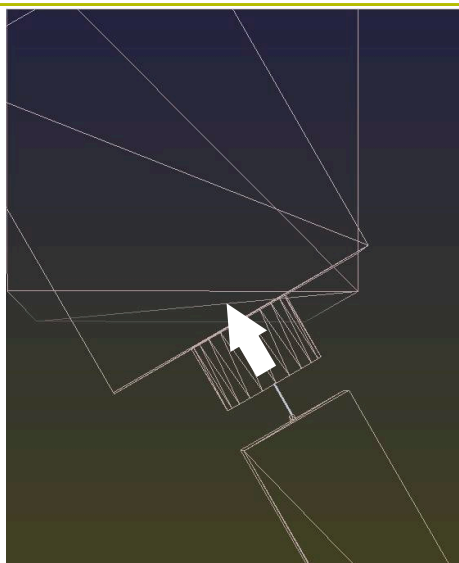
> ℹ️ HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

33

Touch-Probe Cycles for Workpieces (#17 / #1-05-1)

33.1 Overview

Determining workpiece misalignment

Cycle		Call	Further information
400	**BASIC ROTATION** (#17 / #1-05-1)	**DEF**-active	Page 1311
	▪ Automatic measurement using two points		
	▪ Compensation via basic rotation		
401	**ROT OF 2 HOLES** (#17 / #1-05-1)	**DEF**-active	Page 1315
	▪ Automatic measurement using two holes		
	▪ Compensation via basic rotation		
402	**ROT OF 2 STUDS** (#17 / #1-05-1)	**DEF**-active	Page 1320
	▪ Automatic measurement using two studs		
	▪ Compensation via basic rotation		
403	**ROT IN ROTARY AXIS** (#17 / #1-05-1)	**DEF**-active	Page 1325
	▪ Automatic measurement using two points		
	▪ Compensation via rotary table rotation		
404	**SET BASIC ROTATION** (#17 / #1-05-1)	**DEF**-active	Page 1330
	▪ Setting any basic rotation		
405	**ROT IN C AXIS** (#17 / #1-05-1)	**DEF**-active	Page 1331
	▪ Automatic alignment of an angular offset between a hole center and the positive Y axis		
	▪ Compensation via rotary table rotation		
1410	**PROBING ON EDGE** (#17 / #1-05-1)	**DEF**-active	Page 1336
	▪ Automatic measurement using two points		
	▪ Compensation via basic rotation or rotary table rotation		
1411	**PROBING TWO CIRCLES** (#17 / #1-05-1)	**DEF**-active	Page 1342
	▪ Automatic measurement using two holes or studs		
	▪ Compensation via basic rotation or rotary table rotation		
1412	**INCLINED EDGE PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1350
	▪ Automatic measurement using two points on an inclined edge		
	▪ Compensation via basic rotation or rotary table rotation		
1416	**INTERSECTION PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1358
	▪ Automatically determines the intersection with four touch points on two straight lines		
	▪ Compensation via basic rotation or rotary table rotation		
1420	**PROBING IN PLANE** (#17 / #1-05-1)	**DEF**-active	Page 1366

Cycle	Call	Further information
■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation		

Determining a preset

Cycle	Call	Further information
408 **SLOT CENTER PRESET** (#17 / #1-05-1) ■ Measuring the width of an inside slot ■ Setting the slot center as preset	**DEF**-active	Page 1380
409 **RIDGE CENTER PRESET** (#17 / #1-05-1) ■ Measuring the outside width of a ridge ■ Definition of ridge center as preset	**DEF**-active	Page 1385
410 **PRESET INSIDE RECTAN** (#17 / #1-05-1) ■ Measuring the inside length and width of a rectangle ■ Setting the rectangle center as preset	**DEF**-active	Page 1390
411 **PRESET OUTS. RECTAN** (#17 / #1-05-1) ■ Measuring the outside length and width of a rectangle ■ Setting the rectangle center as preset	**DEF**-active	Page 1395
412 **PRESET INSIDE CIRCLE** (#17 / #1-05-1) ■ Measuring any four points on the inside of a circle ■ Setting the circle center as preset	**DEF**-active	Page 1401
413 **PRESET OUTS. CIRCLE** (#17 / #1-05-1) ■ Measuring any four points on the outside of a circle ■ Setting the circle center as preset	**DEF**-active	Page 1407
414 **PRESET OUTS. CORNER** (#17 / #1-05-1) ■ Measuring two straight lines on the outside ■ Setting the intersection of the lines as preset	**DEF**-active	Page 1413
415 **PRESET INSIDE CORNER** (#17 / #1-05-1) ■ Measuring two straight lines on the inside ■ Setting the intersection of the lines as preset	**DEF**-active	Page 1420
416 **PRESET CIRCLE CENTER** (#17 / #1-05-1) ■ Measuring any three holes of a bolt hole circle ■ Definition of the circle center as preset	**DEF**-active	Page 1427
417 **PRESET IN TS AXIS** (#17 / #1-05-1) ■ Measuring the any position in the tool axis ■ Setting any position as preset	**DEF**-active	Page 1433
418 **PRESET FROM 4 HOLES** (#17 / #1-05-1) ■ Measuring two holes on each line crosswise ■ Setting the intersection of the connecting lines as preset	**DEF**-active	Page 1437

Cycle		Call	Further information
419	**PRESET IN ONE AXIS** (#17 / #1-05-1)	**DEF**-active	Page 1442
	■ Measuring any position in a selectable axis		
	■ Definition of any position in a selectable axis as preset		
1400	**POSITION PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1445
	■ Measuring the single position		
	■ Setting as preset, if applicable		
1401	**CIRCLE PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1450
	■ Measuring points on the inside or outside of a circle		
	■ Setting the circle center as preset, if applicable		
1402	**SPHERE PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1455
	■ Measuring points on a sphere		
	■ Definition of sphere center as preset, if necessary		
1404	**PROBE SLOT/RIDGE** (#17 / #1-05-1)	**DEF**-active	Page 1459
	■ Determine the center of a slot width or ridge width		
	■ Set the center as a preset if needed		
1430	**PROBE POSITION OF UNDERCUT** (#17 / #1-05-1)	**DEF**-active	Page 1464
	■ Measuring the undercut		
	■ Measure individual position with L-shaped stylus		
	■ Set the preset if needed		
1434	**PROBE SLOT/RIDGE UNDERCUT** (#17 / #1-05-1)	**DEF**-active	Page 1469
	■ Measuring the undercut		
	■ Measuring the center of the slot width or ridge width with an L-shaped stylus		
	■ Set the center as a preset if needed		

Inspecting the workpiece

Cycle		Call	Further information
0	**REF. PLANE** (#17 / #1-05-1)	**DEF**-active	Page 1482
	■ Measuring a coordinate in a selectable axis		
1	**POLAR PRESET** (#17 / #1-05-1)	**DEF**-active	Page 1484
	■ Measuring a point		
	■ Probing direction via angle		
420	**MEASURE ANGLE** (#17 / #1-05-1)	**DEF**-active	Page 1486
	■ Measuring an angle in the working plane		
421	**MEASURE HOLE** (#17 / #1-05-1)	**DEF**-active	Page 1489
	■ Measuring the position of a hole		
	■ Measuring the diameter of a hole		
	■ Nominal-to-actual value comparison, if applicable		
422	**MEAS. CIRCLE OUTSIDE** (#17 / #1-05-1)	**DEF**-active	Page 1494

Cycle		Call	Further information
	■ Measuring the position of a circular stud		
	■ Measuring the diameter of a circular stud		
	■ Nominal-to-actual value comparison, if applicable		
423	**MEAS. RECTAN. INSIDE** (#17 / #1-05-1)	**DEF**-active	Page 1499
	■ Measuring the position of a rectangular pocket		
	■ Measuring the length and width of a rectangular pocket		
	■ Nominal-to-actual value comparison, if applicable		
424	**MEAS. RECTAN. OUTS.** (#17 / #1-05-1)	**DEF**-active	Page 1504
	■ Measuring the position of a rectangular stud		
	■ Measuring the length and width of a rectangular stud		
	■ Nominal-to-actual value comparison, if applicable		
425	**MEASURE INSIDE WIDTH** (#17 / #1-05-1)	**DEF**-active	Page 1509
	■ Measuring the position of a slot		
	■ Measuring the width of a slot		
	■ Nominal-to-actual value comparison, if applicable		
426	**MEASURE RIDGE WIDTH** (#17 / #1-05-1)	**DEF**-active	Page 1513
	■ Measuring the position of a ridge		
	■ Measuring the width of a ridge		
	■ Nominal-to-actual value comparison, if applicable		
427	**MEASURE COORDINATE** (#17 / #1-05-1)	**DEF**-active	Page 1518
	■ Measuring any coordinate in a selectable axis		
	■ Nominal-to-actual value comparison, if applicable		
430	**MEAS. BOLT HOLE CIRC** (#17 / #1-05-1)	**DEF**-active	Page 1522
	■ Measuring the center point of a bolt hole circle		
	■ Measuring the diameter of a bolt hole circle		
	■ Nominal-to-actual value comparison, if applicable		
431	**MEASURE PLANE** (#17 / #1-05-1)	**DEF**-active	Page 1527
	■ Finding the angle of a plane by measuring three points		

Probing the position in the plane or in space

Cycle		Call	Further information
3	**MEASURING** (#17 / #1-05-1)	**DEF**-active	Page 1535
	■ Touch probe cycle for defining OEM cycles		
4	**MEASURING IN 3-D** (#17 / #1-05-1)	**DEF**-active	Page 1537
	■ Measuring any position		
444	**PROBING IN 3-D** (#17 / #1-05-1)	**DEF**-active	Page 1540

Cycle		Call	Further information
	■ Measuring any position		
	■ Determining the deviation from the nominal coordinates		

Influencing cycle runs

Cycle		Call	Further information
441	**FAST PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1546
	■ Touch probe cycle for defining various touch probe parameters		
1493	**EXTRUSION PROBING** (#17 / #1-05-1)	**DEF**-active	Page 1550
	■ Touch probe cycle for defining an extrusion		
	■ Extrusion direction, length, and number of extrusion points can be programmed		

33.2 Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)

33.2.1 Application
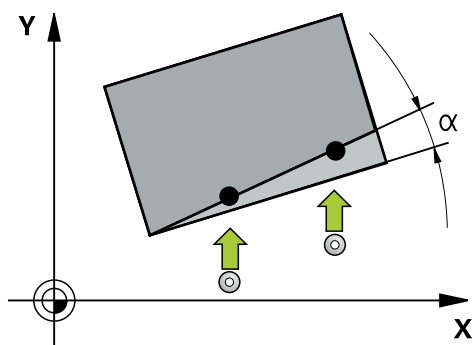
The touch probe cycles contain the following:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3D calibration
- Simultaneous measurement of rotation and position

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

33.2.2 Evaluation

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.

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

Notes

- 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.

33.2.3 Protocol

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.

33.2.4 Notes

- The probing positions are based on the programmed nominal coordinates in the I-CS.
- See your drawing for the nominal positions.
- Before defining a cycle, you must program a tool call in order to define the touch-probe axis.
- The 14xx probing cycles support **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with an L-TYPE stylus, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.
- If the workpiece touch probe does not deflect exactly horizontally or vertically, measuring results may deviate.
- 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.

33.2.5 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".

> **i** Depending on the object, you need to define the nominal positions that determine the probing direction,
> Examples:
> - **Further information:** "Alignment using two holes", Page 1302
> - **Further information:** "Alignment through an edge", Page 1303
> - **Further information:** "Alignment via the plane", Page 1304

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**, then the control opens a message in the **FN 16** tab, **Status** workspace. This message indicates that the mode for traversing to the clearance height is not possible.
> ▶ 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, manually traverse to a clearance height after every 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.
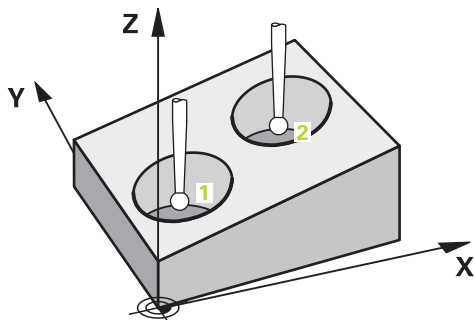
Examples

Important: Specify the **nominal positions** from the drawing!

In the following three examples, the nominal positions from this drawing will be used.
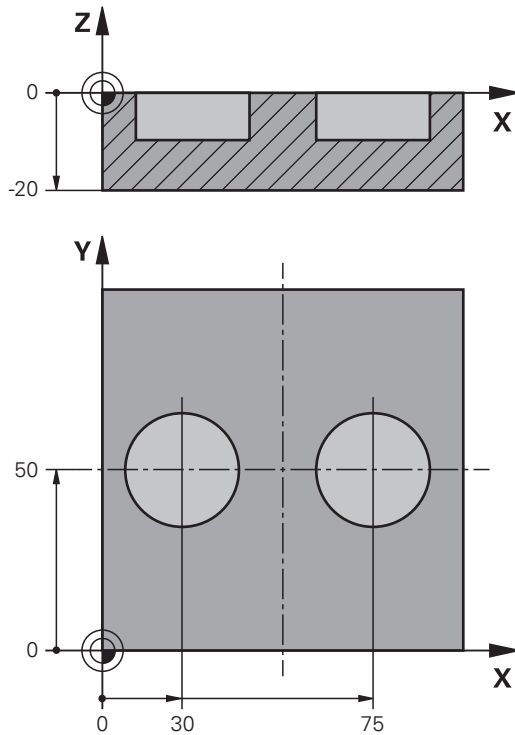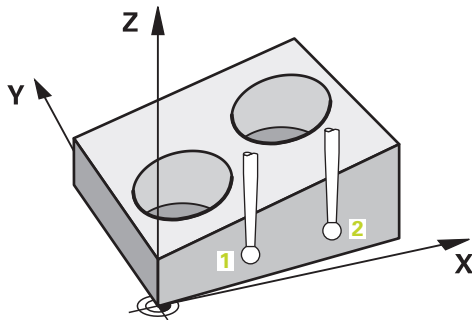
Alignment using two holes

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 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
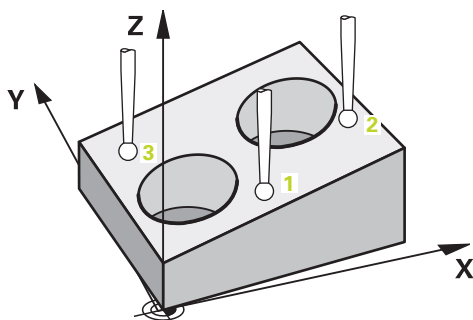- **QS1102** = Nominal Position 1 in tool axis is unknown
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = Nominal Position 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 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 in tool axis is unknown
- **QS1103** = Nominal Position 2 in main axis is unknown
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **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 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 of the tool axis is provided, but the workpiece position is not known
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = Nominal Position 2 of the tool axis is provided, but the workpiece position is not known
- **QS1106** = Nominal Position 3 of the main axis is provided, but the workpiece position is not known
- **QS1107** = Nominal Position 3 of the secondary axis is provided, but the workpiece position is not known
- **QS1108** = Nominal Position 3 of the tool axis is provided, but the workpiece position is not known

11 TCH PROBE 1420 PROBING IN PLANE ~	
QS1100= "?50"	;1ST POINT REF AXIS ~
QS1101= "?10"	;1ST POINT MINOR AXIS ~
QS1102= "?0"	;1ST POINT TOOL AXIS ~
QS1103= "?80"	;2ND POINT REF AXIS ~
QS1104= "?50"	;2ND POINT MINOR AXIS ~
QS1105= "?0"	;2ND POINT TOOL AXIS ~
QS1106= "?20"	;3RD POINT REF AXIS ~
QS1107= "?80"	;3RD POINT MINOR AXIS ~
QS1108= "?0"	;3RD POINT TOOL AXIS ~
Q372=-3	;PROBING DIRECTION ~
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

33.2.6 Evaluation of tolerances

Cycles 14xx also allow you to check tolerance bands. This includes the checking of the position and size of an object.

You can define the following tolerances:

Tolerance	Example
DIN EN ISO 286-2	10H7
ISO 2768-1	10m
Nominal dimension	10+0.01-0.015

You can enter nominal dimensions with the following tolerances:

Combination	Example	Manufacturing dimension
x+-y	10+-0.5	10.0
x-+y	10-+0.5	10.0
x-y+z	10-0.1+0.5	10.2
x+y-z	10+0.1-0.5	9.8
x+y+z	10+0.1+0.5	10.3
x-y-z	10-0.1-0.5	9.7
x+y	10+0.5	10.25
x-y	10-0.5	9.75

If you program a tolerance entry, 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 a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
>   ■ Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

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 the NC program

or

▶ Press **NC Start** to resume NC program run

> ⓘ Please note that the deviations returned by the touch probe cycles are based on the mean tolerance in **Q98x** and **Q99x**. If **Q1120** and **Q1121** are defined, then the values are equivalent to the values used for the compensation. If no automatic evaluation is active, then the control saves the values (based on the mean tolerance) in the intended Q parameter, allowing you to 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

33.2.7 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 the nominal position with tolerance monitoring as needed

▶ 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 based on 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

33.3 Determining workpiece misalignment (#17 / #1-05-1)

33.3.1 Fundamentals of touch probe cycles 400 to 405

Characteristics common to all touch probe cycles for measuring workpiece misalignment
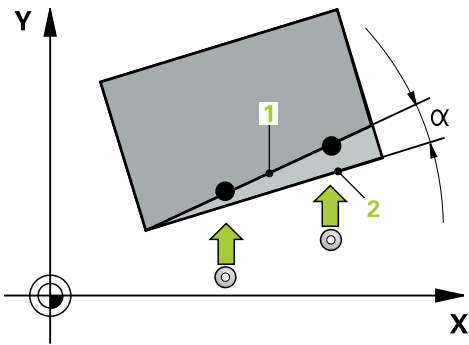
In Cycles **400**, **401**, and **402**, you can use parameter **Q307 Preset value for rotation angle** to define whether the measurement result will be corrected by a known angle α (see figure). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.

ℹ️ These cycles do not work with 3D ROT! In such a case, use cycles **14xx**. **Further information:** "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

33.3.2 Cycle 400 BASIC ROTATION (#17 / #1-05-1)

ISO programming

G400

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.

> **ℹ** Instead of Cycle **400 BASIC ROTATION**, HEIDENHAIN recommends using the more powerful cycles below:
> - **1410 PROBING ON EDGE**
> - **1412 INCLINED EDGE PROBING**

Related topics

- Cycle **1410 PROBING ON EDGE**

  Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)", Page 1336

- Cycle **1412 INCLINED EDGE PROBING**

  Further information: "Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)", Page 1350

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

  Further information: "Positioning logic", Page 235

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
>
> ▶ 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**.
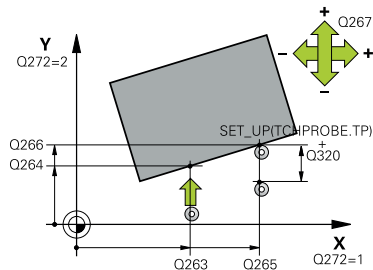> ▶ Reset any coordinate transformations beforehand.

- 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

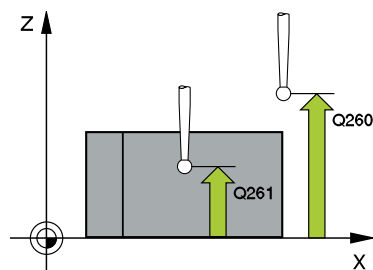| Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This 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. This 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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?

Help graphic	Parameter
	Define how the touch probe will move between the measuring points: **0**: Move to measuring height between measuring points **1**: Move to clearance height between measuring points Input: **0**, **1**
	Q307 Preset value for rotation angle If the misalignment is measured relative to any 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. This 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

33.3.3 Cycle 401 ROT OF 2 HOLES (#17 / #1-05-1)

ISO programming

G401

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 for the calculated value. As an alternative, you can also compensate for the determined misalignment by rotating the rotary table.
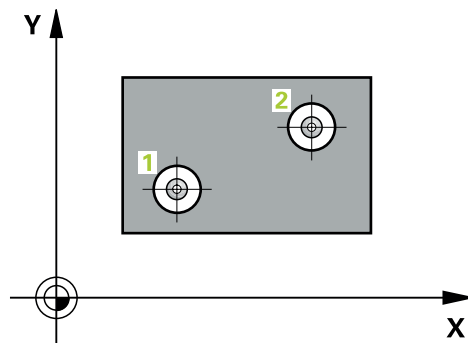
> **i** Instead of Cycle **401 ROT OF 2 HOLES**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

■ Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 1342

Cycle run

1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic

Further information: "Positioning logic", Page 235

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
>
> ▶ 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**.
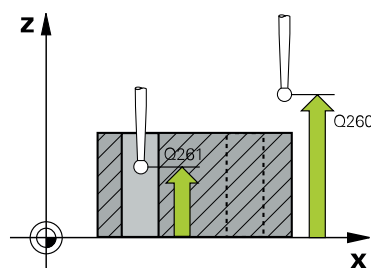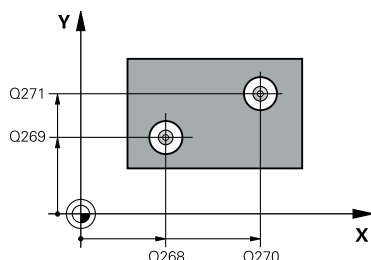> ▶ Reset any coordinate transformations beforehand.

- 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q307 Preset value for rotation angle
	If the misalignment is measured relative to any 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. This 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_OF-FS**). 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

33.3.4 Cycle 402 ROT OF 2 STUDS (#17 / #1-05-1)

ISO programming

G402

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.

> ℹ️ Instead of Cycle **402 ROT OF 2 STUDS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

- Cycle **1411 PROBING TWO CIRCLES**

  Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 1342

Cycle run
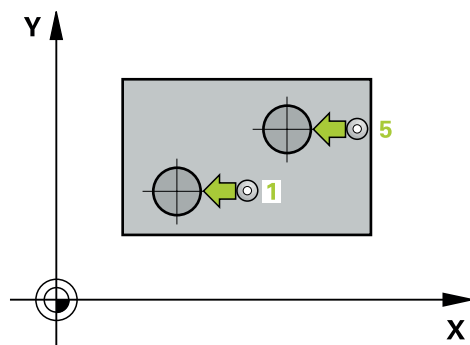
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
>
> ▶ 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**.
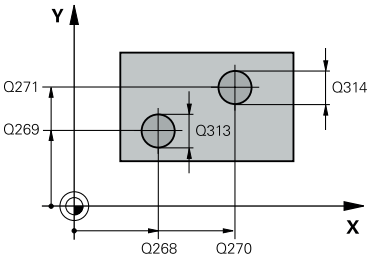>
> ▶ Reset any coordinate transformations beforehand.

- 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?

Help graphic	Parameter
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to clearance height between measuring points
	Input: **0**, **1**

Q307 Preset value for rotation angle

If the misalignment is measured relative to any 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. This 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**

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

Help graphic	Parameter
	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

33.3.5 Cycle 403 ROT IN ROTARY AXIS (#17 / #1-05-1)

ISO programming

G403

Application

Touch probe cycle **403** determines a workpiece misalignment by measuring two points, which must lie on a straight line. The control compensates for the determined misalignment by rotating the A, B, or C axis. The workpiece can be clamped in any position on the rotary table.

> **i** Instead of Cycle **403 ROT IN ROTARY AXIS**, HEIDENHAIN recommends using the more powerful cycles below:
> - **1410 PROBING ON EDGE**
> - **1412 INCLINED EDGE PROBING**

Related topics

- Cycle **1410 PROBING ON EDGE**

  Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)", Page 1336

- Cycle **1412 INCLINED EDGE PROBING**

  Further information: "Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)", Page 1350

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

    Further information: "Positioning logic", Page 235

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 touch points. The calculated angle is in the range of −90° to +90°. There is a risk of collision!

▶ After alignment, check the position of the rotary axis.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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. This 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. This 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. This 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. This 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. This value has an absolute effect. Input: **-99999.9999…+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999…+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?

Help graphic	Parameter
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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**

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
- **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**

Q303 Meas. value transfer (0,1)?

Help graphic	Parameter
	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

33.3.6 Cycle 404 SET BASIC ROTATION (#17 / #1-05-1)

ISO programming

G404

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
>
> ▶ 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

33.3.7 Cycle 405 ROT IN C AXIS (#17 / #1-05-1)

ISO programming

G405

Application
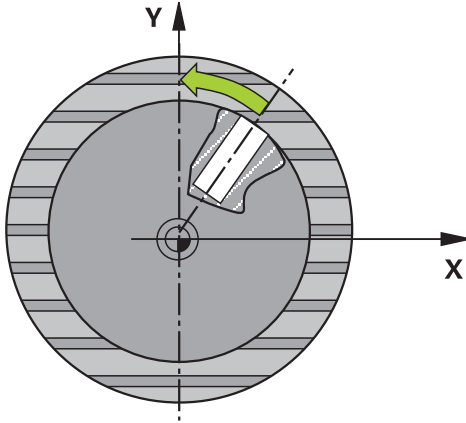
With touch probe cycle **405**, you can measure

- 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 for 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.

> **i** Instead of Cycle **405 ROT IN C AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

- Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 1342

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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. There is a risk of collision!

▶ The pocket/hole must be free of material on the inside

▶ 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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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
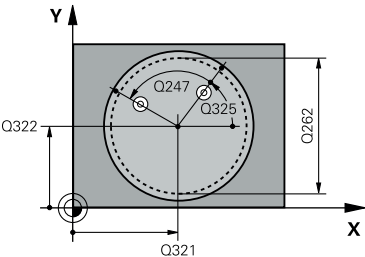
Help graphic	Parameter

Q321 Center in 1st axis?

Center of the hole in the main axis of the working plane. This 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). This 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. This 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Help graphic	Parameter
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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 entered 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

33.3.8 Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)

ISO programming

G1410

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, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1299

- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Further information: "Evaluation of tolerances", Page 1306

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1308

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

  Further information: "Positioning logic", Page 235

2   The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.

3   The control offsets the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.

4   If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

5   The touch probe then moves to the next touch point **2** and probes again.

6   The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined 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 deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
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 ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the second touch point

Notes

NOTICE
Danger of collision!
If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.
▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE
Danger of collision!
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.
  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Note about rotary axes:

- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then 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 tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. 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 for 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 must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 1375

Further information: "Example: Aligning the rotary table from two holes", Page 1377

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 **?, -, +** or **@** ■ **?**: Semi-automatic mode, see Page 1299 ■ **-, +**: Evaluation of the tolerance, see Page 1306 ■ **@**: Transfer of an actual position, see Page 1308
	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. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**

Help graphic	Parameter

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This 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 the clearance height.

0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

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. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

Help graphic	Parameter
	2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.
	3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.
	Input: **0**, **1**, **2**, **3**
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment:
	0: No basic rotation
	1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.
	2: Rotate the rotary table: The control transfers the misalignment to the preset table as an 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

33.3.9 Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)

ISO programming

G1411

Application

Touch probe cycle **1411** captures the centers of two holes or cylindrical studs and calculates a straight line connecting these centers. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1299

- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Further information: "Evaluation of tolerances", Page 1306

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1308

Cycle run

1   The control positions the touch probe to the pre-position of the first touch object **1** at **FMAX** (from the touch probe table), using positioning logic.

Further information: "Positioning logic", Page 235

2   With **FMAX** (from the touch probe table), the touch probe moves to the entered measuring height **Q1102**.

3   Depending on the number of probing processes **Q423**, the touch probe acquires the touch points and ascertains the first hole center or stud center.

4   If you have programmed the **CLEAR. HEIGHT MODE Q1125**, the control will move the touch probe to the clearance height between the touch points and at the end of the probing object. During this process, the control positions the touch probe at **FMAX** from the touch probe table.

5   The control positions the touch probe to the pre-position of the second probing object **2** and repeats steps 2 to 4.

6   After that, the control 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 deviation of the first circle center
Q983 to **Q985**	Measured deviation of the second center
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
	■ **3** = Stylus not moved.
	  The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle.
	  Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546

Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first circle center
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the second circle center
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 1
Q974	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 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, press **NC Start** to execute the cycle with the values from the dialog. The active set-up clearance is reduced to the displayed value only for this object.
■ You can cancel the cycle by pressing Cancel.

Notes

> ## *NOTICE*
>
> **Danger of collision!**
>
> If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.
>
> ▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

> ## *NOTICE*
>
> **Danger of collision!**
>
> When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
>
> ▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.
  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Note about rotary axes:

- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining 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 tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. 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 for 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 must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 1375

Further information: "Example: Aligning the rotary table from two holes", Page 1377

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ■ **"?..."**: Semi-automatic mode, see Page 1299 ■ **"...-...+..."**: Evaluation of the tolerance, see Page 1306 ■ **"...@..."**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** 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, see Page 1306
	Q1103 2nd noml. position of ref axis? Absolute nominal position of the center 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 center 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**)
	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, see Page 1306
	Q1115 Geometry type (0-3)? Type of object to be probed: **0**: Position 1 = hole, and position 2 = hole **1**: Position 1 = stud, and position 2 = stud
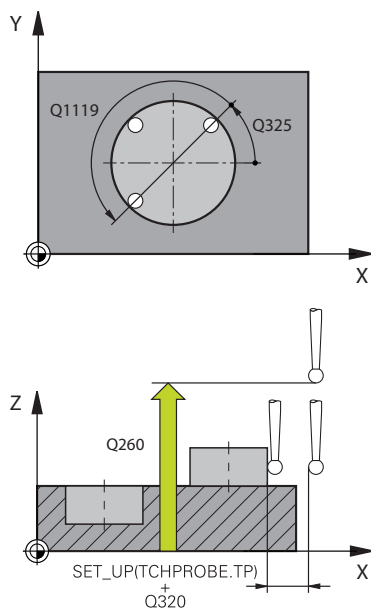
Help graphic	Parameter
	2: Position 1 = hole, and position 2 = stud
	3: Position 1 = stud, and position 2 = hole
	Input: **0**, **1**, **2**, **3**
	Q423 Number of probes?
	Number of touch points on the diameter
	Input: **3**, **4**, **5**, **6**, **7**, **8**
	Q325 Starting angle?
	Angle between the main axis of the working plane and the first touch point. This 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 active 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. This 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 the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. 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. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. **2**: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. **3**: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q1121 CONFIRM ROTATION?
	Define whether the control should use the determined misalignment:
	0: No basic rotation
	1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.
	2: Rotate the rotary table: The control transfers the misalignment to the preset table as an 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

33.3.10 Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)

ISO programming

G1412

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, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.
  Further information: "Semi-automatic mode", Page 1299
- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.
  Further information: "Transferring the actual position", Page 1308

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
  Further information: "Positioning logic", Page 235

2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.

3 The control retracts the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.

4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

5 The touch probe then moves to the touch point **2** and probes again.

6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined 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 deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
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
	■ **3** = Stylus not moved.
	    The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle.
	    Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546

Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the second touch point

Notes

NOTICE
Danger of collision!
If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.
▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE
Danger of collision!
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
▶ 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 **TOLERANCE QS400** parameter to program a tolerance for the surface normal along the inclined edge.
- Observe the fundamentals of touch probe cycles **14xx**.
  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Note about rotary axes:
- When you determine 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 (3D ROT menu) 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 (3D ROT menu) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** in dependence on the tool axis.
- In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines whether the control checks the matching of the tilting situation. If no check is configured, the control 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 for 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 must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 1375

Further information: "Example: Aligning the rotary table from two holes", Page 1377

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 **?**, **+**, **-** or **@** ■ **?**: Semi-automatic mode, see Page 1299 ■ **-, +**: Evaluation of the tolerance, see Page 1306 ■ **@**: Transfer of an actual position, see Page 1308
	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. The control determines this deviation using 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 monitoring of the tolerance. ■ **QS400 ="0"**: No monitoring of the tolerance. ■ **QS400 ="0.1+0.1"** : No monitoring of the tolerance. Input: Max. **255** characters
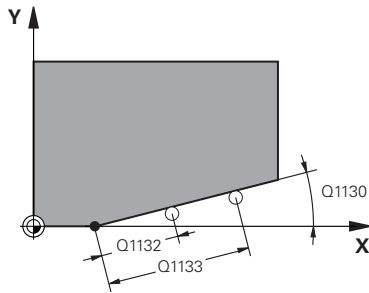
Help graphic	Parameter
	Q1130 Nominal angle for 1st line?
	Nominal angle of the first straight line
	Input: **-180...+180**
	Q1131 Probing direction for 1st line?
	Probing direction for the first edge:
	+1: Rotates the probing direction by +90° to the nominal angle **Q1130** and probes at right angles to the nominal edge.
	-1: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge.
	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 interprets the nominal angle **Q1130** and the probing direction **Q1131**.
	1: YZ plane
	2: ZX plane
	3: XY plane
	Input: **1**, **2**, **3**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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 the clearance height.
	0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.
	1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.
	2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**
	Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error?

Help graphic	Parameter
	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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.
	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. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.

Input: **0**, **1**, **2**, **3**

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation.

2: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset.

Help graphic	Parameter
	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

33.3.11 Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)

ISO programming

G1416

Application

Touch probe cycle **1416** allows you to determine the intersection of two edges. You can execute the cycle in all three machining planes XY, XZ and YZ. The cycle requires a total of four touch points and two positions per edge. You can select the sequence of the edges as desired.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

The cycle also offers the following possibilities:

- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

  Further information: "Semi-automatic mode", Page 1299

- If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

  Further information: "Transferring the actual position", Page 1308

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

    Further information: "Positioning logic", Page 235

2   The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.

3   If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

4   The control positions the touch probe to the next touch point.

5   The control positions the touch probe to the entered measuring height **Q1102** and measures the next touch point.

6   The control repeats Steps 3 to 5 until all four touch points are measured.

7   The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

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
Q959 to **Q960**	Measured intersection in the main axis and secondary axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviation of the first touch point in the main axis, auxiliary axis and tool axis
Q983 to **Q985**	Measured deviation of the second touch point in the main axis, auxiliary axis and tool axis
Q986 to **Q988**	Measured deviation of the third touch point in the main axis, auxiliary axis and tool axis
Q989 to **Q990**	Measured deviations of the intersection in the main axis and secondary axis
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 ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 1st touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 2nd touch point
Q972	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the 3rd touch point

Notes

NOTICE

Danger of collision!

If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.

▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.
  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Note about rotary axes:

- If you determine the basic rotation in a tilted machining plane, then note the following:
  - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining 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 tilting angle (3D-ROT menu) do not match, then the machining plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** based on the tool axis.
- The optional machine parameter **chkTiltingAxes** (no. 204601) allows the machine manufacturer to define whether the control checks for a matching tilting situation. If no check is defined, then the control assumes a consistent machining plane. 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 for 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 must apply the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.
- The alignment with rotary table axes is possible only if no basic rotation was set before.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 1375

Further information: "Example: Aligning the rotary table from two holes", Page 1377

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position in the main axis at which the two edges intersect. Input: **-99999.9999...+99999.9999** or **?** or **@** ■ **?**: Semi-automatic mode, see Page 1299 ■ **@**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis? Absolute nominal position in the secondary axis at which the two edges intersect. Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the touch points in the tool axis Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	QS400 Tolerance value? Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normal along the first edge. The control determines the deviation using the nominal coordinates and the actual coordinates of the part. 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 monitoring of the tolerance. ■ **QS400 ="0"**: No monitoring of the tolerance. ■ **QS400 ="0.1+0.1"** : No monitoring of the tolerance. Input: Max. **255** characters
	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 edge: **+1**: Rotates the probing direction by +90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. **-1**: Rotates the probing direction by -90° to the nominal angle **Q1130** and probes at right angles to the nominal edge. Input: **-1**, **+1**
	Q1132 First distance on 1st line? Distance between the intersection and the first touch point on the first edge. This value has an incremental effect. Input: **-999.999...+999.999**

Help graphic	Parameter
	Q1133 Second distance on 1st line? Distance between the intersection and the second touch point on the first edge. This value has an incremental effect. Input: **-999.999...+999.999**
	QS401 Tolerance value 2? Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normals along the second edge. The control determines this deviation using the nominal coordinate and the actual coordinate of the workpiece. Input: Max. **255** characters
	Q1134 Nominal angle for 2nd line? Nominal angle of the first straight line Input: **-180...+180**
	Q1135 Probing direction for 2nd line? Probing direction for the second edge: **+1**: Rotates the probing direction by +90° relative to the nominal angle **Q1134** and probes at right angles relative to the nominal edge. **-1**: Rotates the probing direction by -90° relative to the nominal angle **Q1134**, and probes at right angles relative to the nominal edge. Input: **-1**, **+1**
	Q1136 First distance on 2nd line? Distance between the intersection and the first touch point on the second edge. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1137 Second distance on 2nd line? Distance between the intersection and the second touch point on the second edge. This value has an incremental effect. Input: **-999.999...+999.999**
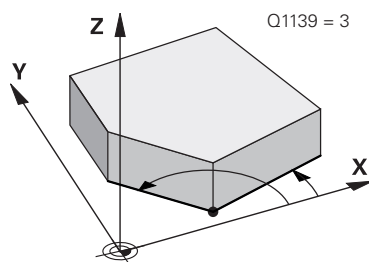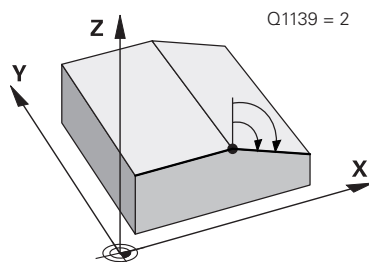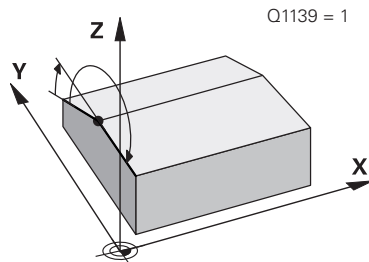
Help graphic	Parameter
Q1139 = 1	**Q1139 Plane for object (1-3)?** Plane in which the control interprets the nominal angle **Q1130** and **Q1134**, as well as the probing direction **Q1131** and **Q1135**. **1**: YZ plane **2**: ZX plane **3**: XY plane Input: **1**, **2**, **3**
Q1139 = 2	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
Q1139 = 3	**Q1125 Traverse to clearance height?** Positioning behavior between the touch points: **-1**: Do not move to the clearance height. **0**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. 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**

Help graphic	Parameter
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the point of intersection. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the intersection. Input: **0**, **1**
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: **0**: No basic rotation **1**: Set the basic rotation: The control transfers the misalignment of the first edge to the preset table as a basic transformation. **2**: Execute rotary table rotation: The control transfers the misalignment of the first edge to the preset table as an offset. **3**: Set the basic rotation: The control transfers the misalignment of the second edge to the preset table as a basic transformation. **4**: Execute rotary table rotation: The control transfers the misalignment of the second edge to the preset table as an offset. **5**: Set basic rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as a basic transformation. **6**: Execute rotary table rotation: The control transfers the misalignment from the mean deviations of both edges to the preset table as an offset. Input: **0**, **1**, **2**, **3**, **4**, **5**, **6**

Example

11 TCH PROBE 1416 INTERSECTION PROBING ~	
Q1100=+50	;1ST POINT REF AXIS ~
Q1101=+10	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS400="0"	;TOLERANCE ~
Q1130=+45	;NOMINAL ANGLE, 1ST LINE ~
Q1131=+1	;PROBE DIRECTION, 1ST LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+25	;SECOND DISTANCE, 1ST LINE ~
QS401="0"	;TOLERANCE 2 ~
Q1134=+135	;NOMINAL ANGLE, 2ND LINE ~
Q1135=-1	;PROBE DIRECTION, 2ND LINE ~
Q1136=+10	;FIRST DISTANCE, 2ND LINE ~
Q1137=+25	;SECOND DISTANCE, 2ND 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

33.3.12 Cycle 1420 PROBING IN PLANE (#17 / #1-05-1)

ISO programming
G1420

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, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

The cycle also offers the following possibilities:

■ If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 1299

■ Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

Further information: "Evaluation of tolerances", Page 1306

■ If you have already determined the exact position beforehand, then you can define the value in the cycle as the nominal position.

Further information: "Transferring the actual position", Page 1308

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

2 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.

3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

4 It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.

5 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.

6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined 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 deviation from the first touch point
Q983 to **Q985**	Measured deviation from the second touch point
Q986 to **Q988**	Third measured deviation of the positions
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle.

Q parameter number	Meaning
	Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the second touch point
Q972	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the third touch point

Notes

NOTICE
Danger of collision!
If, between the objects or touch points, you do not move to a clearance height, then there is a risk of collision.
▶ Move to the clearance height between every object or touch point. Program **Q1125 CLEAR. HEIGHT MODE** so as not to be equal to **-1**.

NOTICE
Danger of collision!
When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
▶ 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 cycle saves the measured spatial angle in parameters **Q961** to **Q963**. For the transfer to the 3D basic rotaton, the control uses the difference between the measured spatial angle and the nominal spatial angle.
- Observe the fundamentals of touch probe cycles **14xx**.
  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

ⓘ	- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!
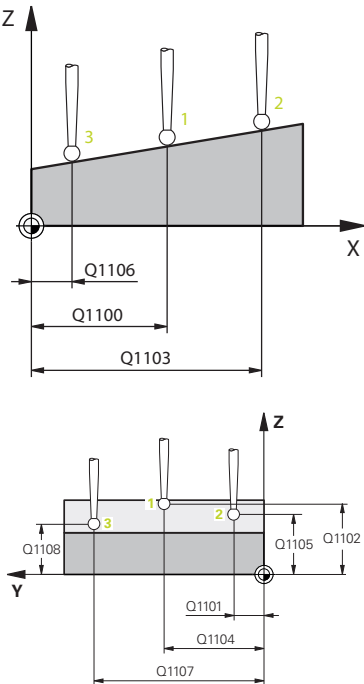
Aligning the rotary table axes:
- Alignment of rotary axes is only possible if two rotary axes are available in the kinematics.
- To align the rotary axes (**Q1126** unequal to 0), the rotation must be accepted (**Q1121** unequal to 0). Otherwise, the control will display an error message.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 1375
Further information: "Example: Aligning the rotary table from two holes", Page 1377

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 **?, -, +** or **@** ■ **?**: Semi-automatic mode, see Page 1299 ■ **-, +**: Evaluation of the tolerance, see Page 1306 ■ **@**: Transfer of an actual position, see Page 1308
	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**)
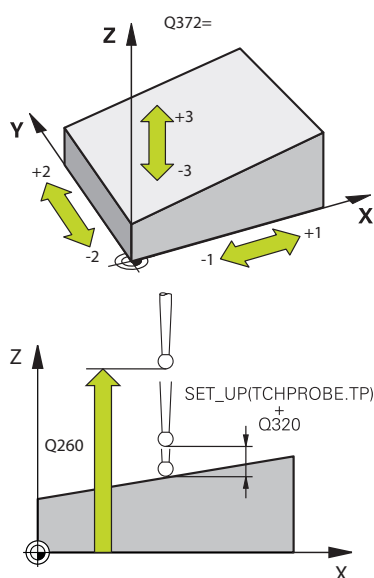
Help graphic	Parameter
	Q1107 3rd noml. position minor axis?
	Absolute nominal position of the third touch point in the secondary axis of the working plane
	Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1108 3rd nominal position tool axis?
	Absolute nominal position of the third 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. The algebraic sign lets you define whether the control moves in the positive or negative direction.
	Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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 the clearance height.
	0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.
	1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.
	2: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.
	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. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.
	2: Correction based on the second touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.
	3: Correction based on 3rd touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 3rd touch point.
	4: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point.
	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 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

33.3.13 Example: Determining a basic rotation from two holes
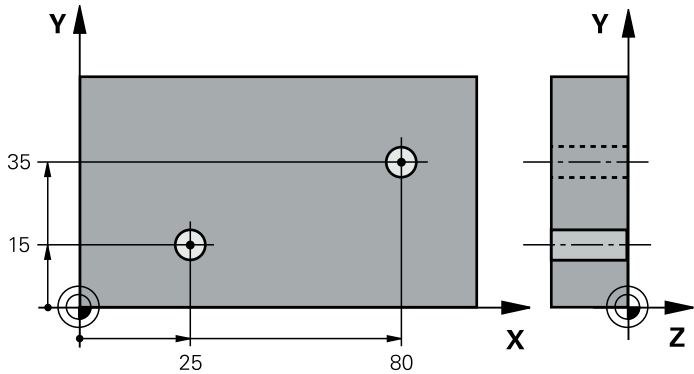
- **Q268** = Center of the 1st hole: X coordinate
- **Q269** = Center of the 1st hole: Y coordinate
- **Q270** = Center of the 2nd hole: X coordinate
- **Q271** = Center of the 2nd hole: Y coordinate
- **Q261** = Coordinate in the touch probe axis in which the measurement is performed
- **Q307** = Angle 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	

33.3.14 Example: Determining a basic rotation from a plane and two holes

When setting a basic rotation with cycles **14xx**, this must be defined by parameters **Q1120 TRANSER POSITION** and **Q1121 CONFIRM ROTATION**.

Program sequence

- Cycle **1420 PROBING IN PLANE**
  - **Q1120=+4**: Compensation to the mean touch point
  - **Q1121=+1**: Set basic rotation
- Cycle **1411 PROBING TWO CIRCLES**
  - **Q1120=+3**: Compensation to the mean touch point
  - **Q1121=+1**: Set basic rotation
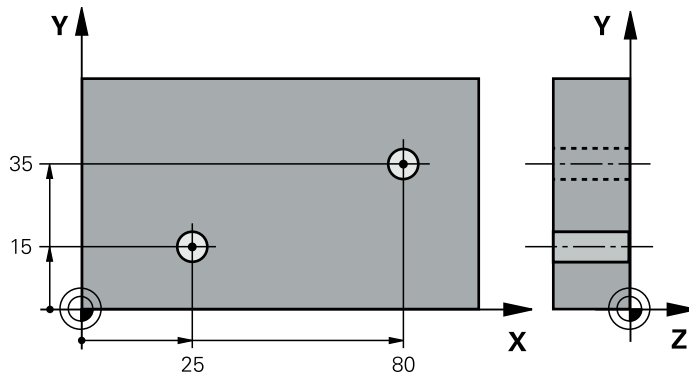
0	BEGIN PGM TOUCHPROBE MM	
1	TOOL CALL 600 Z	
2	TCH PROBE 1420 PROBING IN PLANE ~	
	Q1100=+20	;1ST POINT REF AXIS ~
	Q1101=+20	;1ST POINT MINOR AXIS ~
	Q1102=+0	;1ST POINT TOOL AXIS ~
	Q1103=+80	;2ND POINT REF AXIS ~
	Q1104=+50	;2ND POINT MINOR AXIS ~
	Q1105=+0	;2ND POINT TOOL AXIS ~
	Q1106=+10	;3RD POINT REF AXIS ~
	Q1107=+60	;3RD POINT MINOR AXIS
	Q1108=+0	;3RD POINT TOOL AXIS ~
	Q372=-3	;PROBING DIRECTION ~
	Q320=+2	;SET-UP CLEARANCE ~
	Q260=+50	;CLEAR. HEIGHT MODE ~
	Q1125=+2	;CLEARANCE HEIGHT ~
	Q309=+0	;ERROR REACTION ~
	Q1126=+1	;ALIGN ROTARY AXIS ~
	Q1120=+4	;TRANSER POSITION ~
	Q1121=+1	;CONFIRM ROTATION
3	TCH PROBE 1411 PROBING TWO CIRCLES ~	
	Q1100=+25	;1ST POINT REF AXIS ~
	Q1101=+15	;1ST POINT MINOR AXIS ~
	Q1102=-10	;1ST POINT TOOL AXIS ~

Q1116=+8	;DIAMETER 1 ~	
Q1103=+80	;2ND POINT REF AXIS ~	
Q1104=+35	;2ND POINT MINOR AXIS ~	
Q1105=-10	;2ND POINT TOOL AXIS ~	
Q1117=+8	;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=+50	;CLEARANCE HEIGHT ~	
Q1125=+2	;CLEAR. HEIGHT MODE ~	
Q309=+0	;ERROR REACTION ~	
Q1126=+0	;ALIGN ROTARY AXIS ~	
Q1120=+3	;TRANSER POSITION ~	
Q1121=+1	;CONFIRM ROTATION	
4 CALL PGM 35		; Call the part program
5 END PGM TOUCHPROBE MM		

33.3.15 Example: Aligning the rotary table from two holes

When aligning a rotary table with cycles **14xx**, this must be defined by parameters **Q1126 ALIGN ROTARY AXIS**, **Q1120 TRANSER POSITION** and **Q1121 CONFIRM ROTATION**.

Program sequence

■ Cycle **1411 PROBING TWO CIRCLES**

■ **Q1126=+2**: Position rotary axes with motion control **TURN**

■ **Q1120=+3**: Compensation to the mean touch point

■ **Q1121=+2**: Execute rotary table alignment and accept offset

0	BEGIN PGM TOUCHPROBE MM		
1	TOOL CALL 600 Z		
2	TCH PROBE 1411 PROBING TWO CIRCLES ~		
	Q1100=+25	;1ST POINT REF AXIS ~	
	Q1101=+15	;1ST POINT MINOR AXIS ~	
	Q1102=-10	;1ST POINT TOOL AXIS ~	
	Q1116=+8	;DIAMETER 1 ~	
	Q1103=+80	;2ND POINT REF AXIS ~	
	Q1104=+35	;2ND POINT MINOR AXIS ~	
	Q1105=-10	;2ND POINT TOOL AXIS ~	
	Q1117=+8	;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=+50	;CLEARANCE HEIGHT ~	
	Q1125=+2	;CLEAR. HEIGHT MODE ~	
	Q309=+0	;ERROR REACTION ~	
	Q1126=+2	;ALIGN ROTARY AXIS ~	
	Q1120=+3	;TRANSER POSITION ~	
	Q1121=+2	;CONFIRM ROTATION	
3	CALL PGM 35		; Call the part program
4	END PGM TOUCHPROBE MM		

33.4 Determining the preset (#17 / #1-05-1)

33.4.1 Fundamentals of touch probe cycles 408 to 419 for preset setting

Application

⚙️ Depending on the setting of the optional machine parameter **CfgPresetSettings** (no. 204600), the control will check during probing whether the position of the rotary axis matches the tilting angles **3D ROT**. 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 727

■ **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**

ℹ️ 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.

33.4.2   **Cycle 408** SLOT CENTER PRESET (#17 / #1-05-1)

ISO programming

G408

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.

> **i**   Instead of Cycle **408 SLOT CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

  Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 1459

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

    Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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!

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. There is a risk of collision!

▶ To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot 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 slot in the main axis of the working plane. This 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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: **0**: Move to measuring height between measuring points **1**: Move to clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table?
	Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.
	Further information: "Saving the calculated preset", Page 1378
	Input: **0...99999**
	Q405 New preset?
	Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. This 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. This 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. This 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. This 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. This 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

33.4.3 Cycle 409 RIDGE CENTER PRESET (#17 / #1-05-1)

ISO programming

G409

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.

> **i** Instead of Cycle **409 RIDGE CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

  Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 1459

Cycle run
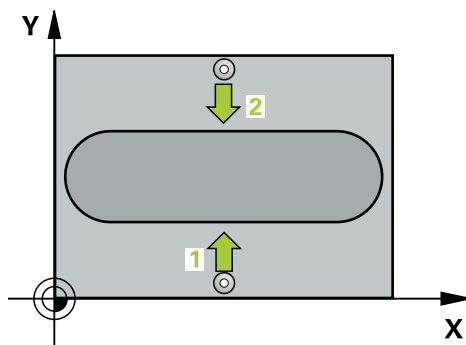
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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
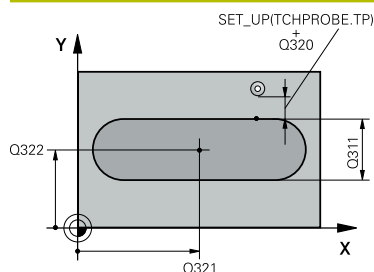
Help graphic	Parameter
	Q321 Center in 1st axis? Center of the ridge in the main axis of the working plane. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the ridge in the secondary axis of the working plane. This value has an absolute effect. 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1378 Input: **0…99999**
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. This 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. This 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. This 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. This value has an absolute effect. Input: **-99999.9999…+99999.9999**
	Q333 New preset in TS axis?

Help graphic	Parameter
	Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This 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

33.4.4   Cycle 410 PRESET INSIDE RECTAN (#17 / #1-05-1)

ISO programming

G410

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 run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

    Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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!

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. There is a risk of collision!

▶ To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second 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 pocket in the main axis of the working plane. This 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. This 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?
	Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to clearance height between measuring points
	Input: **0**, **1**
	Q305 Number in table?
	Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.

Help graphic	Parameter
	Further information: "Saving the calculated preset", Page 1378
	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. This 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. This 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 loaded see "Application", Page 1378
	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. This 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. This 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. This 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. This 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

33.4.5 Cycle 411 PRESET OUTS. RECTAN (#17 / #1-05-1)

ISO programming

G411

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 run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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. This 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. This 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1378 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. This 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. This 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 loaded see "Application", Page 1378 **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. This 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. This 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. This 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. This 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

33.4.6 Cycle 412 PRESET INSIDE CIRCLE (#17 / #1-05-1)

ISO programming

G412

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.

> **i** Instead of Cycle **412 PRESET INSIDE CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

- Cycle **1401 CIRCLE PROBING**

  Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 1450

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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!

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. There is a risk of collision!

▶ The pocket/hole must be free of material on the inside

▶ To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

■ 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:

Help graphic	Parameter
	0: Move to measuring height between measuring points
	1: Move to clearance height between measuring points
	Input: **0**, **1**
	Q305 Number in table?
	Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.
	Further information: "Saving the calculated preset", Page 1378
	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. This 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. This 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 loaded see "Application", Page 1378
	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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis?

Help graphic	Parameter
	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. This 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. This 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. This 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

33.4.7 Cycle 413 PRESET OUTS. CIRCLE (#17 / #1-05-1)

ISO programming

G413

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.

ℹ️ Instead of Cycle **413 PRESET OUTS. CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

- Cycle **1401 CIRCLE PROBING**
  Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 1450

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
    Further information: "Positioning logic", Page 235

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 408 to 419 for preset setting", Page 1378)

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 main 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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to clearance height between measuring points
	Input: **0**, **1**
	Q305 Number in table?
	Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated.
	Further information: "Saving the calculated preset", Page 1378
	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. This 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. This 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 loaded see "Application", Page 1378
	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. This 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. This 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. This 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. This 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

33.4.8 Cycle 414 PRESET OUTS. CORNER (#17 / #1-05-1)

ISO programming

G414

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.

> **i** Instead of Cycle **414 PRESET OUTS. CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

- Cycle **1416 INTERSECTION PROBING**

  Further information: "Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)", Page 1358

Cycle run
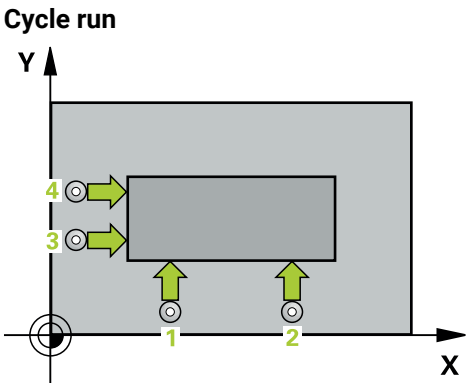
1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

     Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

7   Then the control saves the coordinates of the calculated corner 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.

> 🛈 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).
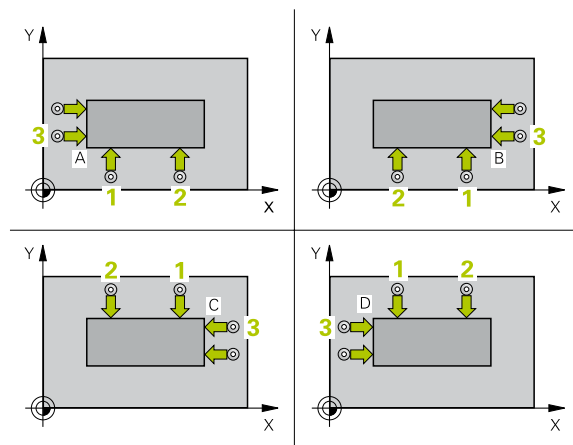
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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 **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 1378
	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. This 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. This 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 loaded see "Application", Page 1378
	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. This 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. This 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. This 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. This 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

33.4.9 Cycle 415 PRESET INSIDE CORNER (#17 / #1-05-1)

ISO programming

G415

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.

> **i** Instead of Cycle **415 PRESET INSIDE CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

■ Cycle **1416 INTERSECTION PROBING**

Further information: "Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)", Page 1358

Cycle run
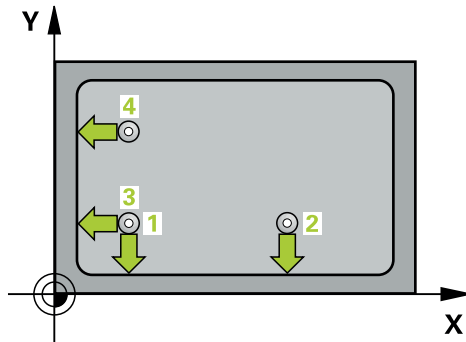
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 235

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 amount of 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 amount of 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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.

9 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

> **i** 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

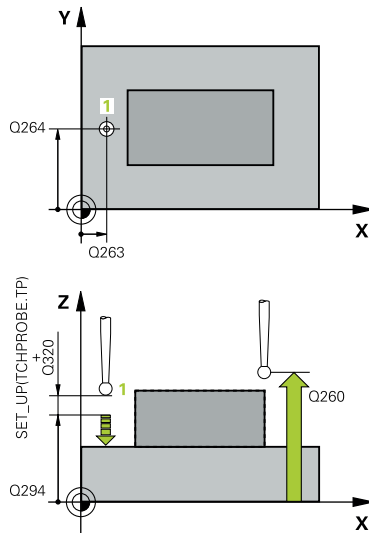| *NOTICE* |
| --- |
| **Danger of collision!** |
| When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision! |
| ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **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. This 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. This 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0**, **1**

Q304 Execute basic rotation (0/1)?

Help graphic	Parameter
	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 **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 1378

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. This 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. This 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 loaded see "Application", Page 1378

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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q383 Probe TS axis: Coord. 2nd axis?

Help graphic	Parameter
	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. This 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. This 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. This 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

33.4.10 Cycle 416 PRESET CIRCLE CENTER (#17 / #1-05-1)

ISO programming

G416

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 run

1   The control positions the touch probe at the entered center of the first hole **1**, using positioning logic

    Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
▶ 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. This 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center 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 **Q303=0**, then the control describes the zero point table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1378

Help graphic	Parameter
	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. This 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. This 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 loaded see "Application", Page 1378

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. This 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. This 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. This 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. This value has an absolute effect.

Input: **-99999.9999…+99999.9999**

Help graphic	Parameter
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active 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

33.4.11 Cycle 417 PRESET IN TS AXIS (#17 / #1-05-1)

ISO programming

G417

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.

> ℹ️ Instead of Cycle **417 PRESET IN TS AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

  Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 1445

Cycle run
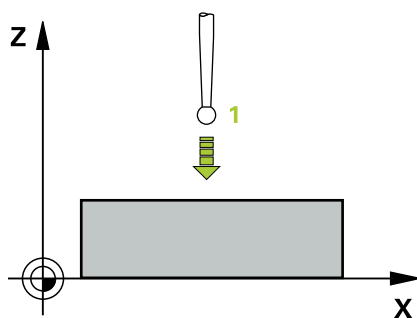
1 Following the positioning logic, the control positions the touch probe to the programmed touch point **1**. In this process, the control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.

   Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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

<table>
<tr><td align="center">*NOTICE*</td></tr>
<tr><td>

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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.

</td></tr>
</table>

- 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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 **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 1378
	Input: **0...99999**
	Q333 New preset in TS axis?
	Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This 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 loaded see "Application", Page 1378
	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.

Help graphic	Parameter
	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

33.4.12 Cycle 418 PRESET FROM 4 HOLES (#17 / #1-05-1)

ISO programming

G418

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 run
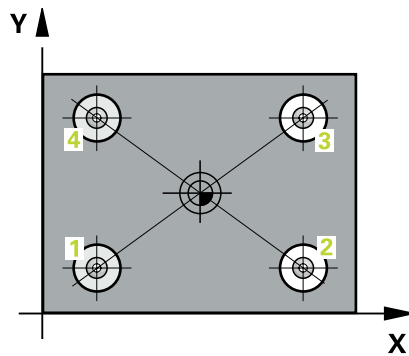
1 The control positions the touch probe at the center of the first hole **1**, using positioning logic

   Further information: "Positioning logic", Page 235

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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 1378)

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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!
▶ 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. This 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. This 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. This 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. This 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q305 Number in table?

Help graphic	Parameter
	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 **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 1378
	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. This 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. This 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 loaded see "Application", Page 1378
	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. This 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. This 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. This 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. This 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

33.4.13 Cycle 419 PRESET IN ONE AXIS (#17 / #1-05-1)

ISO programming

G419

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.

> ℹ️ Instead of Cycle **419 PRESET IN ONE AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

  Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 1445

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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 408 to 419 for preset setting", Page 1378)

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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	Corresponding 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**

Q305 Number in table?

Help graphic	Parameter
	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 **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 1378
	Input: **0…99999**
	Q333 New preset?
	Coordinate at which the control will set the preset. Default setting = 0. This 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 loaded see "Application", Page 1378
	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

33.4.14 Cycle 1400 POSITION PROBING (#17 / #1-05-1)

ISO programming
G1400

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, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.

3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

4 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to **Q982**	Measured deviation from the first touch point
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first touch point

Notes

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.

  Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

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 **?, -, +** or **@**
	■ **?**: Semi-automatic mode, see Page 1299
	■ **-, +**: Evaluation of the tolerance, see Page 1306
	■ **@**: Transfer of an actual position, see Page 1308
	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. The algebraic sign lets you define whether the control moves in the positive or negative direction.
	Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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 the clearance height.
	0, 1, 2: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.
	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. The control corrects the active preset by the amount of deviation between the nominal and actual position of 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

33.4.15 Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)

ISO programming

G1401

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, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point, using positioning logic.

    Further information: "Positioning logic", Page 235

2   The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.

3   If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

4   The control positions the touch probe to the next touch point.

5   The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.

6   Depending on the definition of **Q423 NO. OF PROBE POINTS**, steps 3 to 5 repeat themselves.

7   The control returns the touch probe to the clearance height **Q260**.

8   The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

    Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

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 deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from the first circle center
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation starting from Diameter 1

Notes

<div style="border:1px solid #000;">

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

</div>

■ This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

■ Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ▪ **"?..."**: Semi-automatic mode, see Page 1299 ▪ **"...-...+..."**: Evaluation of the tolerance, see Page 1306 ▪ **"...@..."**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** 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, see Page 1306
	Q1115 Geometry type (0/1)? Type of object to be probed: **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. This 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 active 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. This 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 the clearance height. **0**, **1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. 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. The control corrects the active preset by the amount of deviation between the nominal and actual position of 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

33.4.16 Cycle 1402 SPHERE PROBING (#17 / #1-05-1)

ISO programming

G1402

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 run

1 The control positions the touch probe to the pre-position of the first touch point, using positioning logic.

   Further information: "Positioning logic", Page 235

2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.

3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

4 The control positions the touch probe to the next touch point.

5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.

6 Depending on the definition of **Q423** "Number of probe measurements", steps 3 to 5 repeat themselves.

7 The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.

8 The touch probe moves to the center of the sphere and probes another touch point.

9 The touch probe returns to the clearance height **Q260**.

10 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to **Q982**	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546

Notes

> ### *NOTICE*
>
> **Danger of collision!**
>
> When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
>
> ▶ 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**.
■ Observe the fundamentals of touch probe cycles **14xx**.
   Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis?
	Absolute nominal position of the center in the main axis of the working plane.
	Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**:
	■ **"?..."**: Semi-automatic mode, see Page 1299
	■ **"...-...+..."**: Evaluation of the tolerance, see Page 1306
	■ **"...@..."**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis?
	Absolute nominal position of the center in the secondary axis of the working plane
	Input: **-99999.9999...+9999.9999** 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
	Input: **0...9999.9999** or optional input (see **Q1100**)
	■ **"...-...+..."**: Evaluation of the tolerance, see Page 1306
	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. This 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 active 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. This 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 the clearance height.
	0, **1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.
	2: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.
	Input: **0**, **1**, **2**
	Q1120 Transfer position?
	Define which touch point will be used to correct the active preset:
	0: No correction
	1: Correction of the active preset based on the center of the sphere. The control corrects the active present by the amount of the deviation of the nominal and actual position of the center.
	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

33.4.17 Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)

ISO programming
G1404

Application

Touch probe cycle **1404** determines the center of the width of a slot or ridge. The control probes the two opposing touch points. The control probes perpendicularly to the angle of rotation of the object to be probed, even if the object to be probed is rotated. You can apply the result to the active row of the preset table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

Cycle run
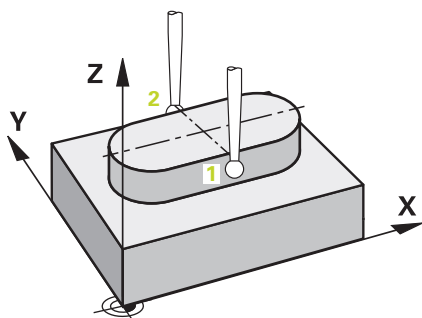
1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

2   The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.

3   Depending on the selected type of geometry in the parameter **Q1115**, the control proceeds as follows:

   Slot **Q1115=0**:

   ■   If you program **CLEAR. HEIGHT MODE Q1125** with the value **0**, **1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to **Q260 CLEARANCE HEIGHT**.

   Ridge **Q1115=1**:

   ■   Independently of **Q1125**, the control positions the touch probe at **FMAX_PROBE** after every touch point back to **Q260 CLEARANCE HEIGHT**.

4   The touch probe moves to the next touch point **2** and performs the second probing procedure at the probing rate **F**.

5   The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Q parameter number	Meaning
Q950 to **Q952**	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to **Q982**	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation from the center of the slot or ridge
Q975	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the slot width or ridge width

Notes

> **NOTICE**
>
> **Danger of collision!**
>
> When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
>
> ▶ Reset any coordinate transformations before the cycle call.

■ This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

■ Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ▪ **"?..."**: Semi-automatic mode, see Page 1299 ▪ **"...-...+..."**: Evaluation of the tolerance, see Page 1306 ▪ **"...@..."**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the touch points in the tool axis Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1113 Width of slot/ridge? Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect. Input: **0...9999.9999** Or **-** or **+**: ▪ **"...-...+..."**: Evaluation of the tolerance, see Page 1306
	Q1115 Geometry type (0/1)? Type of object to be probed: **0**: Slot **1**: Ridge Input: **0**, **1**
	Q1114 Angle of rotation? Angle about which the slot or the ridge is rotated. The center of rotation is in **Q1100** and **Q1101**. This value has an absolute effect. Input: **0...359999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q1125 Traverse to clearance height?

Q1125 Traverse to clearance height?

Positioning behavior between the touch points with a slot:

-1: Do not move to the clearance height.

0, 1: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.

2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**.

The parameter takes effect only with **Q1115=+1** (slot).

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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

Input: **0**, **1**, **2**

Q1120 Transfer position?

Define which touch point will be used to correct the active preset:

0: No correction

1: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center.

Input: **0**, **1**

Example

11 TCH PROBE 1404 PROBE SLOT/RIDGE ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;GEOMETRY TYPE ~
Q1114=+0	;ANGLE OF ROTATION ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

33.4.18 Cycle 1430 PROBE POSITION OF UNDERCUT (#17 / #1-05-1)

ISO programming

G1430

Application

Touch probe cycle **1430** allows a position to be probed with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. You can apply the result of the probing procedure to the active rows of the preset table.

In the main axis and secondary axis, the touch probe is oriented in accordance with the calibration angle. In the tool axis, the touch probe is oriented in accordance with the programmed spindle angle and the calibration angle.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

Cycle run
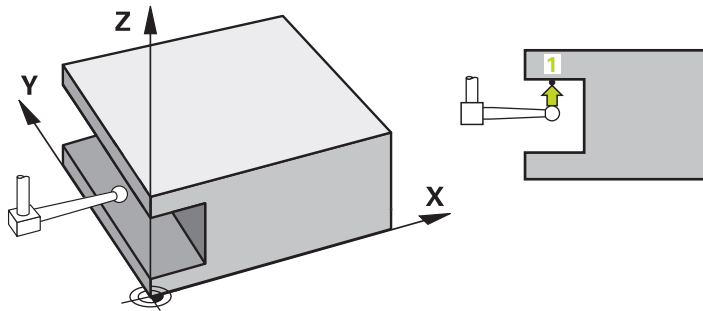
1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Pre-position in the machining plane based on the probing direction:

   ■ **Q372=+/-1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1100**. The radial approach length takes effect in the direction opposite to the probing direction.

   ■ **Q372=+/-2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the **Q1101**. The radial approach length takes effect in the direction opposite to the probing direction.

   ■ **Q372=+/-3**: The pre-position of the main axis and secondary axis depends on the direction in which the stylus is oriented. The pre-position is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position. The radial approach length takes effect in the direction opposite to the spindle angle **Q336**.

   Further information: "Positioning logic", Page 235

2   The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.

3   The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.

4   If you program **CLEAR. HEIGHT MODE Q1125** with the value **0**, **1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

5   The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Q parameter number	Meaning
Q950 to **Q952**	Measured position in the main axis, auxiliary axis and tool axis
Q980 to **Q982**	Measured deviation of the position in the main axis, auxiliary axis and tool axis
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the nominal position of the first touch point

Notes

> **NOTICE**
>
> **Danger of collision!**
>
> When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.
>
> ▶ Reset any coordinate transformations before the cycle call.

■ This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

■ This cycle is not intended for L-shaped styli. For simple styli, HEIDENHAIN recommends Cycle **1400 POSITION PROBING**.

Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 1445

■ Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

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 **?, -, +** or **@** ■ **?**: Semi-automatic mode, see Page 1299 ■ **-, +**: Evaluation of the tolerance, see Page 1306 ■ **@**: Transfer of an actual position, see Page 1308
	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. The algebraic sign lets you define whether the control moves in the positive or negative direction. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	Q336 Angle for spindle orientation? Angle at which the control orients the tool prior to the probing procedure. This angle takes effect only during probing in the tool axis (**Q372 = +/- 3**). This value has an absolute effect. Input: **0...360**
	Q1118 Distance of radial approach? Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. If **Q372= +/-1**: Distance is in the direction opposite to the probing direction. If **Q372= +/- 2**: Distance is in the direction opposite to the probing direction. If **Q372= +/-3**: Distance is in the direction opposite to the angle of the spindle **Q336**. This value has an incremental effect. Input: **0...9999.9999**
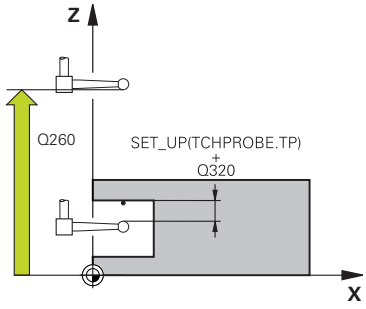
Help graphic	Parameter

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This 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 the clearance height.

0, 1, 2: Move to the 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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level.

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. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point.

Input: **0**, **1**

Example

11 TCH PROBE 1430 PROBE POSITION OF UNDERCUT ~	
Q1100=+10	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-15	;1ST POINT TOOL AXIS ~
Q372=+1	;PROBING DIRECTION ~
Q336=+0	;ANGLE OF SPINDLE ~
Q1118=+20	;RADIAL APPROACH PATH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

33.4.19 Cycle 1434 PROBE SLOT/RIDGE UNDERCUT (#17 / #1-05-1)

ISO programming

G1434

Application

Touch probe cycle **1434** determines the center and width of a slot or a ridge using an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes the two opposing touch points. You can apply the result to the active row of the preset table.

The control orients the touch probe to the calibration angle from the touch probe table.

If, prior to this cycle, you program Cycle **1493 EXTRUSION PROBING**, then the control repeats the touch points in the selected direction and at the defined length along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 1550

Cycle run
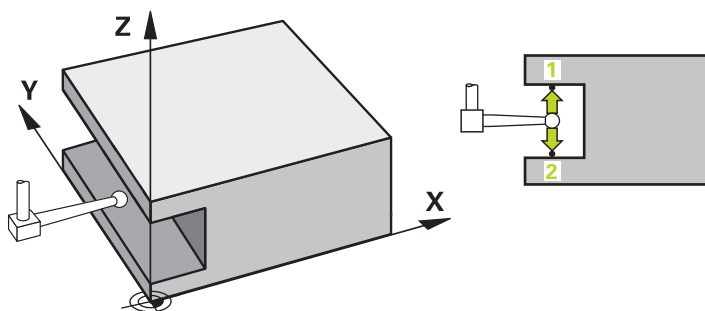
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

The pre-position in the machining plane depends on the object plane:

- **Q1139=+1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1100**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the secondary axis is equivalent to the nominal position.

- **Q1139=+2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position in **Q1101**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the main axis is equivalent to the nominal position.

Further information: "Positioning logic", Page 235

2 The control then positions the touch probe at the entered measuring height **Q1102** and performs the first probing procedure **1** at probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.

3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.

4 The control positions the touch probe to the next touch point **2** and performs the second probing procedure at probing feed rate **F**.

5 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.

6 If you program the parameter **CLEAR. HEIGHT MODE Q1125** with the value **0** or **1**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSER POSITION** is defined with the value **1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Q parameter number	Meaning
Q950 to **Q952**	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to **Q982**	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap ■ **3** = Stylus not moved. The control displays the workpiece status **3** only in connection with the **441 FAST PROBING** cycle. **Further information:** "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 1546
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the center of the slot or the ridge
Q975	If you have programmed Cycle **1493 EXTRUSION PROBING**: Maximum deviation based on the slot width or ridge width

Notes

NOTICE

Danger of collision!

When touch probe cycles **444** and **14xx** are executed, the following coordinate transformation must not be active: Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, Cycle **26 AXIS-SPECIFIC SCALING** and **TRANS MIRROR**. There is a risk of collision.

▶ Reset any coordinate transformations before the cycle call.

■ This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

■ If you program in the radial approach length **Q1118=-0**, then the algebraic sign has no effect. The behavior is identical to +0.

■ This cycle is intended for an L-shaped stylus. For simple styli, HEIDENHAIN recommends Cycle **1404 PROBE SLOT/RIDGE**.

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 1459

■ Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 1297

Cycle parameters

Help graphic	Parameter
	Q1100 1st noml. position of ref. axis? Absolute nominal position of the center in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or enter **?**, **+**, **-** or **@**: ■ **"?..."**: Semi-automatic mode, see Page 1299 ■ **"...-...+..."**: Evaluation of the tolerance, see Page 1306 ■ **"...@..."**: Transfer of an actual position, see Page 1308
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute spindle position of the center in the tool axis Input: **-99999.9999...+9999.9999** Optional input (see **Q1100**)
	Q1113 Width of slot/ridge? Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect. Input: **0...9999.9999** Or **-** or **+**: **"...-...+..."**: Evaluation of the tolerance, see Page 1306
	Q1115 Geometry type (0/1)? Type of object to be probed: **0**: Slot **1**: Ridge Input: **0**, **1**
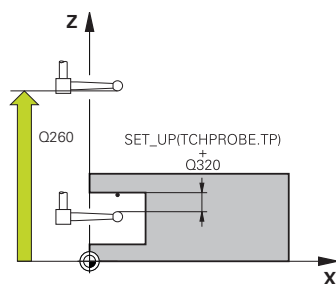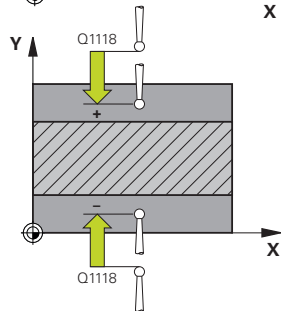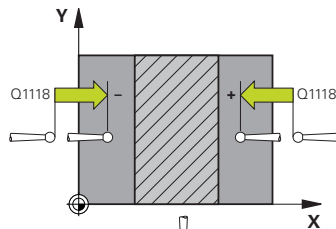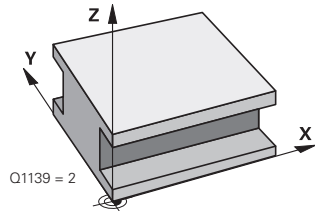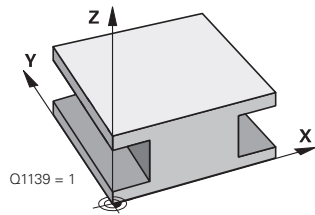
Help graphic	Parameter
	Q1139 Object plane (1-2)? Plane in which the control interprets the probing direction. **1**: YZ plane **2**: ZX plane Input: **1**, **2**
	Q1118 Distance of radial approach? Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. The direction of **Q1118** is equivalent to the probing direction and is in the direction opposite to the algebraic sign. This value has an incremental effect. Input: **-99999.9999...+9999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior before and after the cycle: **-1**: Do not move to the clearance height. **0, 1**: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**
	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 does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction of the active preset based on the center of the slot or the ridge. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center. Input: **0**, **1**

Example

11 TCH PROBE 1434 PROBE SLOT/RIDGE UNDERCUT ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;GEOMETRY TYPE ~
Q1139=+1	;OBJECT PLANE ~
Q1118=-15	;RADIAL APPROACH PATH ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSER POSITION

33.4.20 Example: Presetting at center of a circular segment and on top surface of workpiece
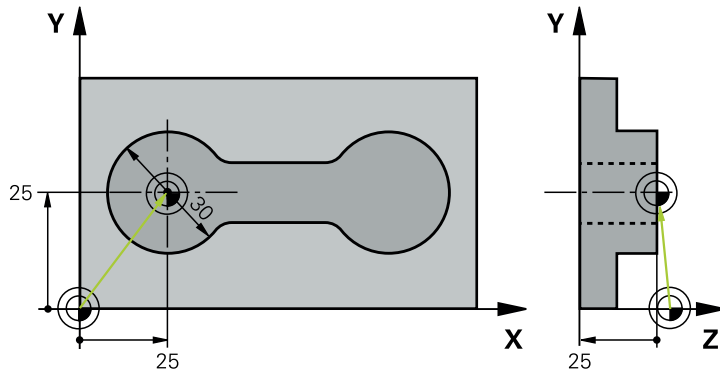
- ■ **Q325** = Polar coordinate angle for touch point 1
- ■ **Q247** = Stepping angle for calculating the touch points 2 to 4
- ■ **Q305** = Write to row number 5 of the preset table
- ■ **Q303** = Write the calculated preset to the preset table
- ■ **Q381** = Also set the preset in the touch probe axis
- ■ **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		

33.4.21 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.
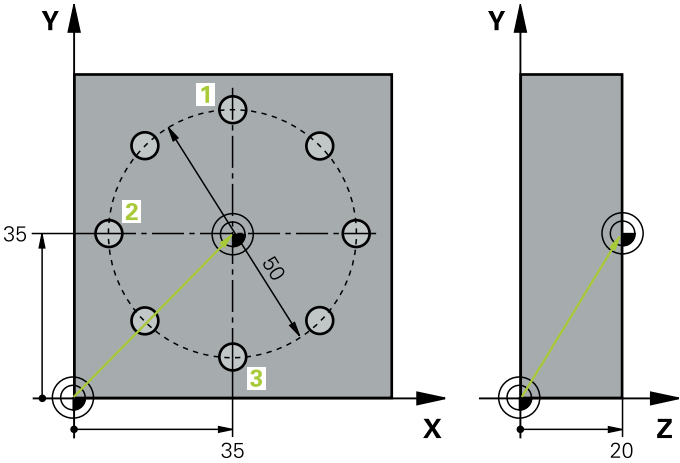
- ■ **Q291** = Polar coordinate angle for first hole center **1**
- ■ **Q292** = Polar coordinate angle for second hole center **2**
- ■ **Q293** = Polar coordinate angle for third hole center **3**

- **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	

33.5 Checking the workpiece (#17 / #1-05-1)

33.5.1 Fundamentals of touch probe cycles 0, 1 and 420 to 431

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.

> **i** 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.

> **i** 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.

> **i** ■ 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 compensates for as follows:

The control always compensates for 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 poll whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Compensating for 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"

■ **FN 18: 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 256

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.

33.5.2 Cycle 0 REF. PLANE (#17 / #1-05-1)

ISO programming

G55

Application

The touch probe cycle measures any position on the workpiece in a selectable axis direction.

> **i** Instead of Cycle **0 REF. PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

  Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 1445

Cycle run
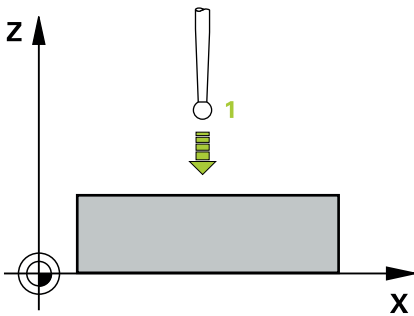
1 In a 3D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.

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 3D 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

33.5.3 Cycle 1 POLAR PRESET (#17 / #1-05-1)

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **1** measures any position on the workpiece in any probing direction.
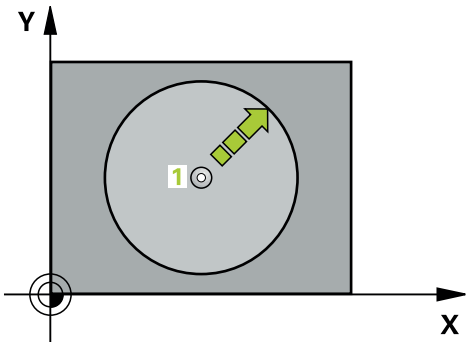
Cycle sequence

1 In a 3D 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 3D 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 ANGLE:+30
13 TCH PROBE 1.2 X+0 Y+10 Z+3

33.5.4 Cycle 420 MEASURE ANGLE (#17 / #1-05-1)

ISO programming

G420

Application

Touch probe cycle **420** measures the angle that any straight line on the workpiece forms with the main axis of the working plane.

> ℹ️ Instead of Cycle **420 MEASURE ANGLE**, HEIDENHAIN recommends using the more powerful Cycle **1410 PROBING ON EDGE**.

Related topics

- Cycle **1410 PROBING ON EDGE**

  Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)", Page 1336

Cycle run
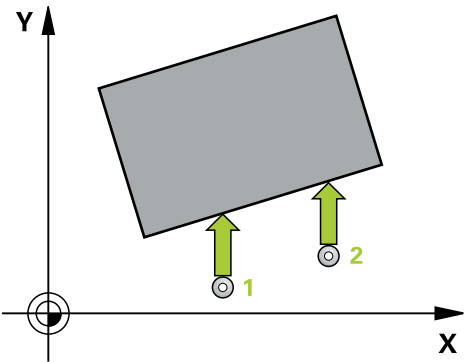
1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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 main 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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

33.5.5 Cycle 421 MEASURE HOLE (#17 / #1-05-1)

ISO programming

G421

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.

> **i** Instead of Cycle **421 MEASURE HOLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

- Cycle **1401 CIRCLE PROBING**

  Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 1450

Cycle run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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.
- The nominal diameter **Q262** must be between the minimum and maximum dimension (**Q276**/**Q275**).
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

Help graphic	Parameter
	Q273 Center in 1st axis (nom. value)?
	Center of the hole in the main axis of the working plane. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Number or name of the tool used for machining. Via 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
	Further information: "Tool monitoring", Page 1480
	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

Help graphic	Parameter
	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**
	Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Example

11 TCH PROBE 421 MEASURE HOLE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+15.25	;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=+15.34	;MAXIMUM LIMIT ~
Q276=+15.16	;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

33.5.6 Cycle 422 MEAS. CIRCLE OUTSIDE (#17 / #1-05-1)

ISO programming

G422

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.

> **i** Instead of Cycle **422 MEAS. CIRCLE OUTSIDE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

- Cycle **1401 CIRCLE PROBING**

   Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 1450

Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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.
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

Help graphic	Parameter

Q273 Center in 1st axis (nom. value)?

Center of the stud in the main axis of the working plane. This 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?
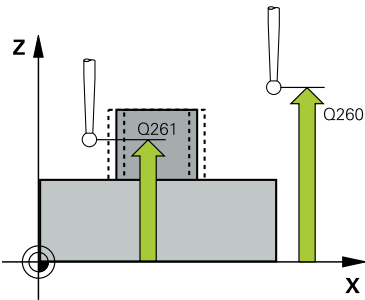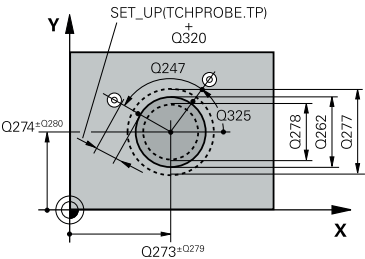
Enter the diameter of the stud.

Input: **0...99999.9999**

Q325 Starting angle?

Angle between the main axis of the working plane and the first touch point. This 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0**, **1**

Q277 Maximum limit of size for stud?

Help graphic	Parameter
	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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Tool number in tool table TOOL.T
	Input: **0...99999.9** or max. **255** characters
	Further information: "Tool monitoring", Page 1480
	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.

Help graphic	Parameter
	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**

Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

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

33.5.7 Cycle 423 MEAS. RECTAN. INSIDE (#17 / #1-05-1)

ISO programming

G423

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.
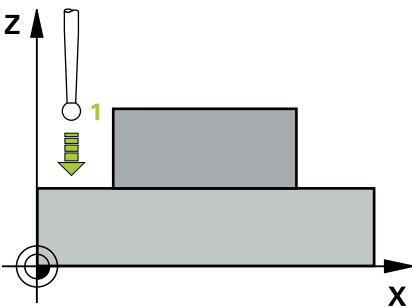
Cycle run

1   The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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**
	Q286 Max. size limit 2nd side length?
	Maximum permissible width for the pocket

Help graphic	Parameter
	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 should perform tool monitoring:

0: Monitoring not active

> 0: Tool number in tool table TOOL.T

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 1480

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

33.5.8 Cycle 424 MEAS. RECTAN. OUTS. (#17 / #1-05-1)

ISO programming

G424

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 run

1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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**
	Q286 Max. size limit 2nd side length?
	Maximum permissible width for the stud

Help graphic	Parameter
	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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Number or name of the tool used for machining. Via 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
	Further information: "Tool monitoring", Page 1480

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

33.5.9 Cycle 425 MEASURE INSIDE WIDTH (#17 / #1-05-1)

ISO programming

G425

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.

> **i** Instead of Cycle **425 MEASURE INSIDE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

  Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 1459

Cycle run
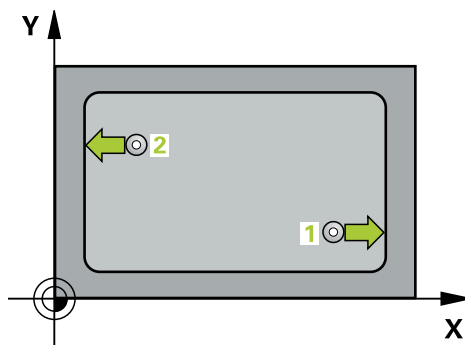
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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
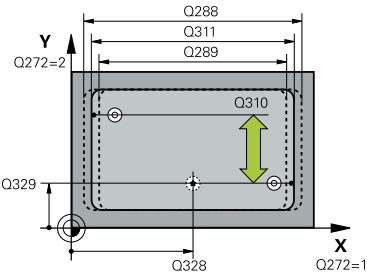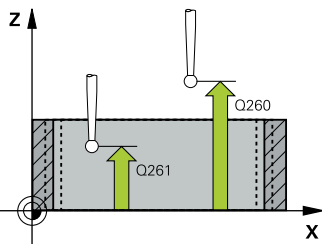
Notes

- 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.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal length **Q311** must be between the minimum and maximum dimension (**Q276**/**Q275**).

Cycle parameters

Help graphic	Parameter
	Q328 Starting point in 1st axis?
	Starting point for probing in the main axis of the working plane. This 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. This 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. This 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. This 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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Number or name of the tool used for machining. Via 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
	Further information: "Tool monitoring", Page 1480
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only active 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)?
	Define how the touch probe will move between the measuring points:
	0: Move to measuring height between measuring points
	1: Move to 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

33.5.10 Cycle 426 MEASURE RIDGE WIDTH (#17 / #1-05-1)

ISO programming

G426

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.

> **i** Instead of Cycle **426 MEASURE RIDGE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

■ Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 1459

Cycle run
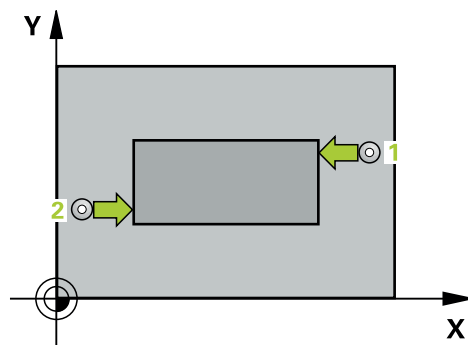
1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 235

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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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?

Help graphic	Parameter
	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 should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via 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

Further information: "Tool monitoring", Page 1480

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

33.5.11 Cycle 427 MEASURE COORDINATE (#17 / #1-05-1)

ISO programming

G427

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.

> **i** Instead of Cycle **427 MEASURE COORDINATE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

  Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 1445

Cycle run

1. The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

   Further information: "Positioning logic", Page 235

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.

■ The measuring height **Q261** must be between the minimum and maximum dimension (**Q276**/**Q275**).

■ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive 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. This 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. This 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. This value has an absolute effect.
	Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance?
	Additional distance between touch point and ball tip. **Q320** is active 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. This 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 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**

Help graphic	Parameter
	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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Number or name of the tool used for machining. Via 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
	Further information: "Tool monitoring", Page 1480
	Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

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

33.5.12 Cycle 430 MEAS. BOLT HOLE CIRC (#17 / #1-05-1)

ISO programming

G430

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 run

1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic

   Further information: "Positioning logic", Page 235

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. This 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. This 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. This 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. This 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. This 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. This 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. This 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**
	Q280 Tolerance for center 2nd axis?

Help graphic	Parameter
	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 should perform tool monitoring:
	0: Monitoring not active
	> 0: Number or name of the tool used for machining. Via 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
	Further information: "Tool monitoring", Page 1480

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

33.5.13 Cycle 431 MEASURE PLANE (#17 / #1-05-1)

ISO programming

G431

Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

> ℹ️ Instead of Cycle **431 MEASURE PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1420 PROBING IN PLANE**.

Related topics

- Cycle **1420 PROBING IN PLANE**

  Further information: "Cycle 1420 PROBING IN PLANE (#17 / #1-05-1)", Page 1366

Cycle run
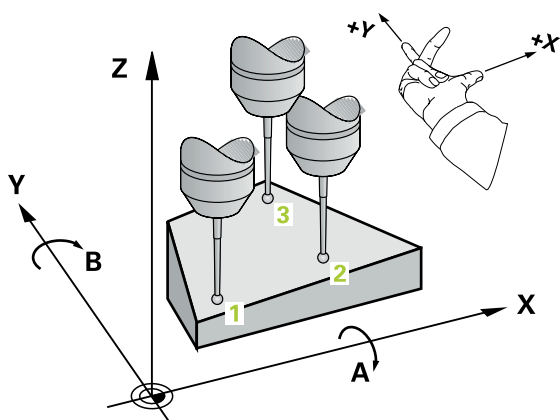
1 The control positions the touch probe to the programmed touch point **1**, using positioning logic and measures the first plane point there. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.

Further information: "Positioning logic", Page 235

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

Risk 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. There is a risk of collision!

▶ 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

<table>
<tr><td rowspan="9">

</td><td>

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q294 1st measuring point in 3rd axis?

Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q295 2nd measuring point in 3rd axis?

Coordinate of the second touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q296 3rd measuring point in 1st axis?

Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q297 3rd measuring point in 2nd axis?

Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
<tr><td>

Q298 3rd measuring point in 3rd axis?

Coordinate of the third touch point in the touch probe axis. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**
</td></tr>
</table>

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active 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. This value has an absolute effect.

Help graphic	Parameter
	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

33.5.14 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
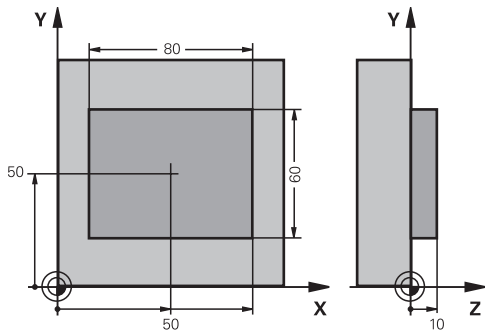
0	BEGIN PGM TOUCHPROBE MM		
1	**TOOL CALL 5 Z S6000**	; Tool call: roughing
2	**Q1 = 81**	; Rectangle length in X (roughing dimension)
3	**Q2 = 61**	; Rectangle length in Y (roughing dimension)
4	**L Z+100 R0 FMAX M3**	; Retract the tool
5	**CALL LBL 1**	; Call the subprogram for machining
6	**L Z+100 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
14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	; End of program
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	

33.5.15 Example: Probing a rectangular pocket and recording the results
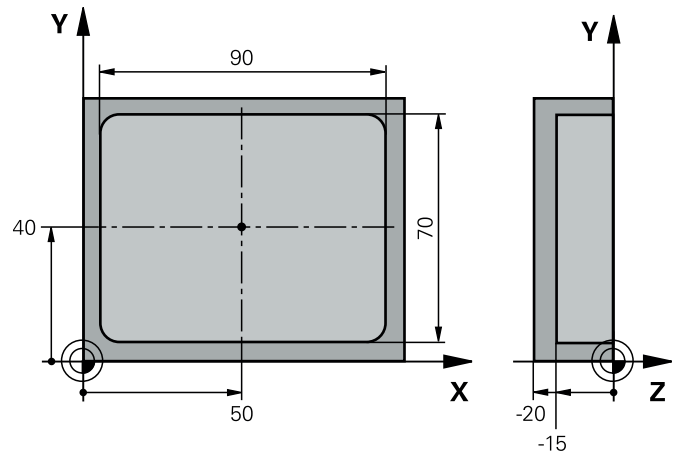
0	BEGIN PGM TOUCHPROBE_2 MM		
1	TOOL CALL 600 Z		; Tool call: touch probe
2	L Z+100 R0 FMAX		; Retract the touch probe
3	TCH PROBE 423 MEAS. RECTAN. INSIDE ~		
	Q273=+50	;CENTER IN 1ST AXIS ~	
	Q274=+40	;CENTER IN 2ND AXIS ~	
	Q282=+90	;FIRST SIDE LENGTH ~	
	Q283=+70	;2ND SIDE LENGTH ~	
	Q261=-5	;MEASURING 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
5	M30		; End of program
6	END PGM TOUCHPROBE_2 MM		

33.6 Probing a position in the plane or in space (#17 / #1-05-1)

33.6.1 Cycle 3 MEASURING (#17 / #1-05-1)

ISO programming

NC syntax is available only in Klartext programming.

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 manufacturer 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 an 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.

> ℹ The **FN 17: SYSWRITE ID990 NR6** function allows setting whether the cycle runs 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?
	This angle defines the probing direction. The angle refers to the probe axis. Confirm with the **ENT** key.
	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: **0...999999999**
	Feed rate measurement
	Enter the measuring feed rate in mm/min.
	Input: **0...3000**
	Maximum retraction distance?
	Traverse path in the direction opposite to 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

33.6.2 Cycle 4 MEASURING IN 3-D (#17 / #1-05-1)

ISO programming

NC syntax is available only in Klartext programming.

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.

> **i** 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! There is a danger of collision!
▶ 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

33.6.3 Cycle 444 PROBING IN 3-D (#17 / #1-05-1)

ISO programming
G444

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
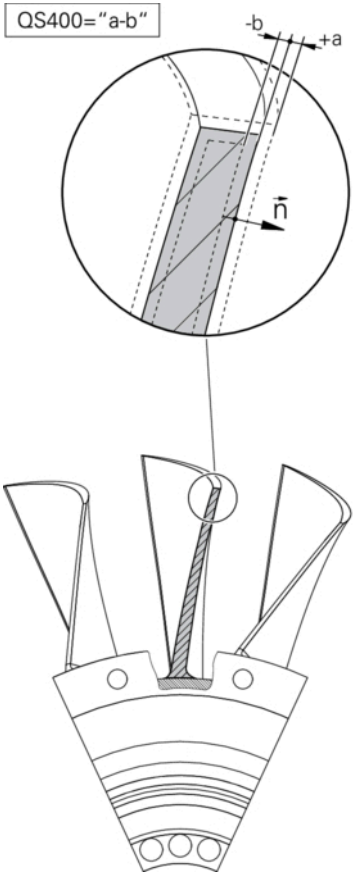
Cycle **444** checks one specific point on the surface of a part. This cycle is used, for example, to measure free-form surfaces of moldmaking parts. It can be determined whether a point on the surface of the part lies in an undersize or oversize range compared to a nominal coordinate. The operator can 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 run
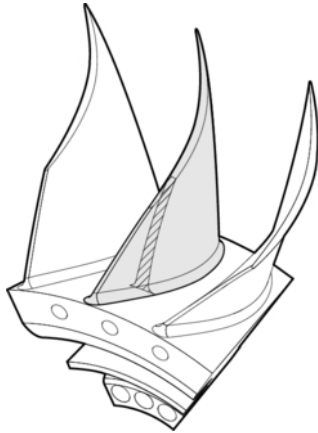
1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table\tchprobe.tp) + **Q320**. Pre-positioning takes a clearance height into account.

    Further information: "Executing touch probe cycles", Page 233

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 3D 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 3D error. 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 **FN 16**).

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 3D calibration before executing Cycle **444**. 3D calibration requires **3D-ToolComp**.
- 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 angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters.
  Further information: "Application", Page 1540
- The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309**.
  Further information: "Application", Page 1540

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. This 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. This 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. This 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 active 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. This 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

33.7 Influencing cycle runs (#17 / #1-05-1)

33.7.1 Cycle 441 FAST PROBING (#17 / #1-05-1)

ISO programming

G441

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.

> **i** In this cycle, no machine movements will be performed.

Program interruption Q400=1

Parameter **Q400 INTERRUPTION** allows interrupting the cycle run and displaying the obtained results.

Program interruption by **Q400** is effective in the following touch probe cycles:

- Touch probe cycles for checking the workpiece: **421** to **427**, **430** and **431**
- Cycle **444 PROBING IN 3-D**
- Touch probe cycles for measuring the kinematics: **45x**
- Touch probe cycles for calibrating: **46x**
- Touch probe cycles **14xx**

Cycles 421 to 427, 430 and 431:

The control displays the results obtained during a program interruption in a **FN 16** monitor output.

Cycles 444, 45x, 46x, 14xx:

The control automatically shows the results obtained during a program interruption in an HTML log in the path: **TNC:\TCHPRlast.html**. You can open the HTML log in the **Document** workspace.

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 manufacturer 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.
- If **Q371** is unequal to **0** and the stylus does not move in cycles **14xx**, the control will terminate the cycle. The control returns the touch probe to the clearance height and saves the workpiece status **3** in Q parameter **Q183**. The NC program continues.

  Workpiece status **3**: Stylus does not move

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:
	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**
	Further information: "Program interruption Q400=1", Page 1546
	Q371 Touch point not reached?
	Define how the control behaves when the stylus does not move within the **DIST** value of the touch probe table.
	0: The control interrupts the NC program with an error message saying that the touch point cannot be reached. This is standard behavior.
	1: The control displays a warning and terminates the probing cycle. The NC program continues. Is effective only in the **14xx** cycles.
	2: The control displays no warning and terminates the probing cycle. The NC program continues. Is effective only in the **14xx** cycles.
	Input: **0**, **1**, **2**

Example

11 TCH PROBE 441 FAST PROBING ~	
Q396=+3000	;POSITIONING FEEDRATE ~
Q397=+0	;SELECT FEED RATE ~
Q399=+1	;ANGLE TRACKING ~
Q400=+1	;INTERRUPTION ~
Q371=+0	;TOUCH POINT REACTION

33.7.2 Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)

ISO programming
G1493

Application
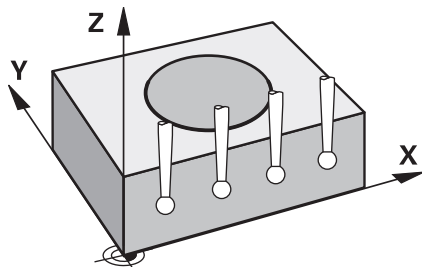
Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

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**, ISO: **G1420**) (#17 / #1-05-1), see Page 1366
- **PROBING ON EDGE** (Cycle **1410**, ISO: **G1410**) (#17 / #1-05-1), see Page 1336
- **PROBING TWO CIRCLES** (Cycle **1411**, ISO: **G1411**) (#17 / #1-05-1), see Page 1342
- **INCLINED EDGE PROBING** (Cycle **1412**, ISO: **G1412**) (#17 / #1-05-1), see Page 1350
- **INTERSECTION PROBING** (Cycle **1416**, ISO: **G1416**) (#17 / #1-05-1), see Page 1358
- **POSITION PROBING** (Cycle **1400**, ISO: **G1400**) (#17 / #1-05-1), see Page 1445
- **CIRCLE PROBING** (Cycle **1401**, ISO: **G1401**) (#17 / #1-05-1), see Page 1450
- **PROBE SLOT/RIDGE** (Cycle **1404**, ISO: **G1404**) (#17 / #1-05-1), see Page 1459
- **PROBE POSITION OF UNDERCUT** (Cycle **1430**, ISO: **G1430**) (#17 / #1-05-1), see Page 1464
- **PROBE SLOT/RIDGE UNDERCUT** (Cycle **1434**, ISO: **G1434**) (#17 / #1-05-1), see Page 1469

Result parameter Q

The control saves the results of the touch probe 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

Result parameter QS

The control saves the individual results of all measuring points of an extrusion in the QS parameters **QS97x**. The result is ten characters long. The results are separated from each other by a space.

Example: **QS970 = 0.12345678 -1.1234567 -2.1234567 -3.1234567**

QS parameter number	Meaning
QS970	Results of touch point 1 of an extrusion
QS971	Results of touch point 2 of an extrusion
QS972	Results of touch point 3 of an extrusion
QS973	Results of diameter 1 of an extrusion
QS974	Results of diameter 2 of an extrusion

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

A touch probe cycle produces the following results within QS parameter **QS970**:

QS970 = 0.12345678 -1.1234567

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS970 BEG0 LEN10)	; Read out the first result from **QS970**
12 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL0**
13 QS0 = SUBSTR (SRC_QS970 BEG11 LEN10)	; Read out the second result from **QS970**
14 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL2**

Further information: "String functions", Page 1048

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3D 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:

  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.
- When executing an extrusion with Cycles **1401 CIRCLE PROBING**, **1411 PROBING TWO CIRCLES** or **1404 PROBE SLOT/RIDGE**, the extrusion direction must equal **Q1140=+3**, otherwise the control will produce an error message.
- When defining the **TRANSER POSITION Q1120>0** within a touch probe cycle, the control will compensate the preset by the mean of deviations. The control calculates this mean from all measured extrusion points of the probing object according to the programmed **TRANSER POSITION Q1120**.

  Example:
  - Nominal position of touch point 1: 2.35 mm
  - Results: **QS970** = 2.30000000 2.35000000 2.40000000 2.50000000

    Mean: 2.387500000 mm

    The preset is corrected by the mean from the nominal position, in this case by 0.0375 mm.

Cycle parameters
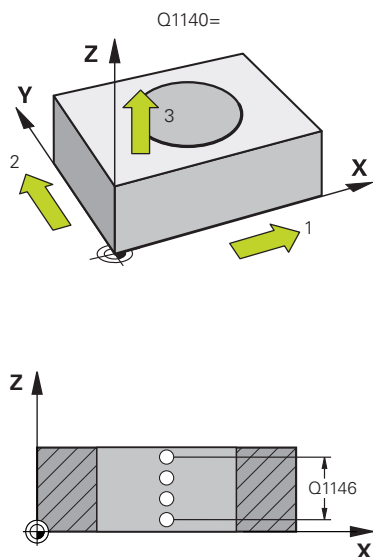
Help graphic	Parameter
	Q1140 Direction for extrusion (1-3)? **1:** Extrusion in the direction of the main axis **2:** Extrusion in the direction of the secondary axis **3:** Extrusion in the direction of the tool axis Input: **1**, **2**, **3**
	Q1145 Number of extrusion points? Number of measuring 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

34

Touch-Probe
Cycles for Tools
(#17 / #1-05-1)

34.1 Overview

Measurement of milling cutters

Cycle		Call	Further information
481	**CAL. TOOL LENGTH** (#17 / #1-05-1) ■ Measuring the tool length	**DEF**-active	Page 1562
482	**CAL. TOOL RADIUS** (#17 / #1-05-1) ■ Measuring the tool radius	**DEF**-active	Page 1565
483	**MEASURE TOOL** (#17 / #1-05-1) ■ Measuring the tool length and radius	**DEF**-active	Page 1569

34.2 Fundamentals

34.2.1 Application

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

Related topics

■ Calibrate the tool touch probe
   Further information: "Calibrating a tool touch probe (#17 / #1-05-1)", Page 1249

34.2.2 Measuring a tool of length 0

> ⚙ Refer to your machine manual!
> The optional machine parameter **maxToolLengthTT** (no. 122607) enables the machine manufacturer to define a maximum tool length for the tool measurement cycles.

> ⓘ HEIDENHAIN recommends that you always define tools with their actual tool length if possible.

The tool measuring cycles measure tools automatically. You can also measure tools defined with a length **L** of 0 in the tool table. To do this, the machine manufacturer must define a maximum tool length value in the optional machine parameter **maxToolLengthTT** (no. 122607). The control starts a search in which the actual tool length is roughly determined in the first step. This is followed by a fine measurement.

Cycle run

1 The tool travels to a clearance height centered above the touch probe.

   The clearance height equals the value of the optional machine parameter **maxToolLengthTT** (no. 122607).

2 The control performs a rough measurement with the spindle standing still.

   When measuring a stationary tool, the control will use the feed rate for probing defined in the machine parameter **probingFeed** (no. 122709).

3 The control saves the roughly measured length.

4 The control performs a fine measurement with the values from the tool measuring cycle.

Notes

NOTICE

Risk of collision!

If the machine manufacturer fails to define the optional machine parameter **maxToolLengthTT** (no. 122607), there will be no tool search. The control pre-positions the tool with a length of 0. Risk of collision!

▶ Observe the machine parameter value in the machine manual.

▶ Define tools with the actual tool length **L**

NOTICE

Risk of collision!

Risk of collision if the tool is longer than the value of the optional machine parameter **maxToolLengthTT** (no. 122607)!

▶ Observe the machine parameter value in the machine manual

34.2.3 Setting machine parameters

⚙ ■ The touch probe cycles **480**, **481**, **482**, **483**, **484** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

ℹ 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).

Setting of the spindle speed

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

Abbreviation	Definition
n	Shaft speed [rpm]
maxPeriphSpeedMeas	Maximum permissible cutting speed in m/min
r	Active tool radius [mm]

Setting of the feed rate

The probing feed rate is calculated as follows:

v = measuring tolerance · n

Abbreviation	Definition
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. The control provides the following options:

- **ConstantTolerance**
- **VariableTolerance**
- **ConstantFeed**

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.

- **VariableTolerance**:

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**

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

Abbreviation	Definition
r	Active tool radius [mm]
measureTolerance1	Maximum permissible error of measurement

Setting for consideration of parallel axes and changes in the kinematics

> ⚙ Refer to your machine manual.
> Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Auxiliary or parallel axes cannot be probed, regardless of the setting of the optional machine parameter **calPosType** (no. 122606).

If the machine manufacturer changes the setting of the optional machine parameter, you need to recalibrate the tool touch probe.

34.2.4 Entries in the tool table for milling tools

Abbr.	Inputs	Dialog
CUT	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	**Number of teeth?**
LTOL	Permitted tool length deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...5.0000**	**Wear tolerance: length?**
RTOL	Permitted tool radius deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...5.0000**	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: **-**, **+**	**Cutting direction (M3 = -)?**
R-OFFS	Position of tool upon length measurement, offset between the probe contact center and the tool center for automatic tool measurement. Default setting: No value entered (offset = tool radius) Input: **-99999.9999...+99999.9999**	**Tool offset: radius?**
L-OFFS	Position of tool upon radius measurement, distance between the probe contact top edge and the tool tip for automatic tool measurement. Is added to the **offsetToolAxis** machine parameter (no. 122707). Input: **-99999.9999...+99999.9999**	**Tool offset: length?**
LBREAK	Permitted tool length deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...9.0000**	**Breakage tolerance: length?**
RBREAK	Permitted tool radius deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column **TL** (status **L**). Input: **0.0000...9.0000**	**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.

34.3 Measurement of milling cutters (#17 / #1-05-1)

34.3.1 Cycle 481 CAL. TOOL LENGTH (#17 / #1-05-1)

ISO programming
G481

Application

> ⚙ Refer to your machine manual!

For measuring the tool length, program touch probe cycle **482**. Via input parameters you can measure the length of a tool by three methods:

- 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.

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**.
- Cycle **481** supports neither turning tools nor dressing tools nor touch probes.

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**
	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

11 TOOL CALL 12 Z	
12 TCH PROBE 481 CAL. TOOL LENGTH ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;PROBING THE TEETH

34.3.2   Cycle 482 CAL. TOOL RADIUS (#17 / #1-05-1)

ISO programming
G482

Application

> ⚙️   Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **482**. Select via input parameters by which of two methods the tool radius is to be measured:

- 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.

Further information: "Notes for individual tooth measurement Q341=1", Page 1566

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.
- Cycle **482** supports neither turning tools nor dressing tools nor touch probes.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manu-facturer 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 tooth 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.

Notes for individual tooth measurement Q341=1

> ### *NOTICE*
>
> **Caution: Danger to the tool and workpiece!**
>
> Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.
>
> ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
>
> ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 - a\tan\left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}}\right)$$

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool

> ⓘ On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

> ### *NOTICE*
>
> **Caution: Possible material damage!**
>
> Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.
>
> ▶ Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

1 Angle deviation

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

11 TOOL CALL 12 Z	
12 TCH PROBE 482 CAL. TOOL RADIUS ~	
Q340=+1	**;CHECK ~**
Q260=+100	**;CLEARANCE HEIGHT ~**
Q341=+1	**;PROBING THE TEETH**

34.3.3 Cycle 483 MEASURE TOOL (#17 / #1-05-1)

ISO programming
G483

Application

> ⚙ Refer to your machine manual!

To measure the tool completely (length and radius), program touch probe cycle **483**. 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 **481** and **482**.

Further information: "Notes for individual tooth measurement of radius Q341=1", Page 1571

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.
- Cycle **483** supports neither turning tools nor dressing tools nor touch probes.

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 tooth 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.

Notes for individual tooth measurement of radius Q341=1

> ### *NOTICE*
>
> **Caution: Danger to the tool and workpiece!**
>
> Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.
>
> ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
> ▶ Check the workpiece optically in order to exclude broken tools

---

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 - atan\left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}}\right)$$

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool

> ⓘ On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

> ### *NOTICE*
>
> **Caution: Possible material damage!**
>
> Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.
>
> ▶ Check the workpiece dimensions during subsequent machining operations

---

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.

120°

1 Angle deviation

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

11 TOOL CALL 12 Z	
12 TCH PROBE 483 MEASURE TOOL ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;PROBING THE TEETH

35

Touch-Probe Cycles for Kinematics Measuring

35.1 Overview

Cycle		Call	Further information
450	**SAVE KINEMATICS** (#17 / #1-05-1) and (#48 / #2-01-1)	**DEF**-active	Page 1580
	▪ Storing the active machine kinematic configuration		
	▪ Restoring previously saved kinematic configuration		
451	**MEASURE KINEMATICS** (#17 / #1-05-1) and (#48 / #2-01-1)	**DEF**-active	Page 1583
	▪ Automatic checking of the machine kinematic configuration		
	▪ Optimizing the machine kinematic configuration		
452	**PRESET COMPENSATION** (#17 / #1-05-1) and (#48 / #2-01-1)	**DEF**-active	Page 1599
	▪ Automatic checking of the machine kinematic configuration		
	▪ Optimizing the kinematic transformation chain of the machine		

35.2 Fundamentals (#48 / #2-01-1)

35.2.1 Fundamentals
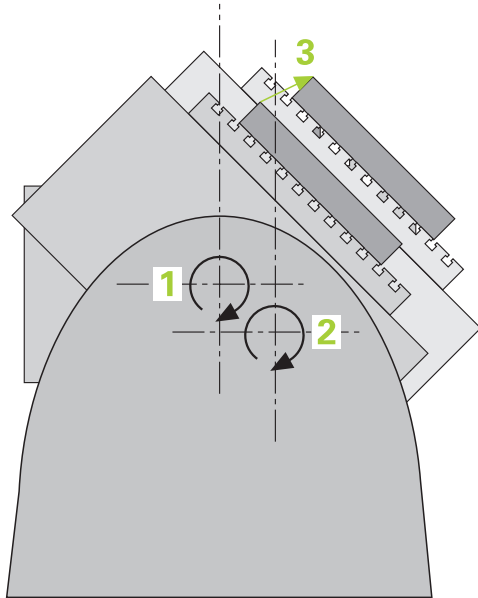
Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

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 meet these complex requirements in real life: a 3D 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.

35.2.2 Requirements

> ⚙ Refer to your machine manual.
>
> The Advanced Function Set 1 software option (#8 / #1-01-1) must have been enabled.
>
> Software option (#48 / #2-01-1) must have been enabled.
>
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Requirements for using KinematicsOpt:

> ⚙ The machine manufacturer 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 manufacturer and is used to position the rotary axes

- The 3D 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 manufacturer during commissioning)

> ℹ HEIDENHAIN recommends using the calibration spheres **KKH 250 (ordering number: 655475-01)** or **KKH 80 (ordering 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.

35.2.3 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**, all cycles for coordinate transformation must be inactive. There is a danger of collision!

▶ 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.

35.3   Storing, measuring and optimizing kinematics (#48 / #2-01-1)

35.3.1   Cycle 450 SAVE KINEMATICS (#48 / #2-01-1)

ISO programming
G450

Application

> ⚙   Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
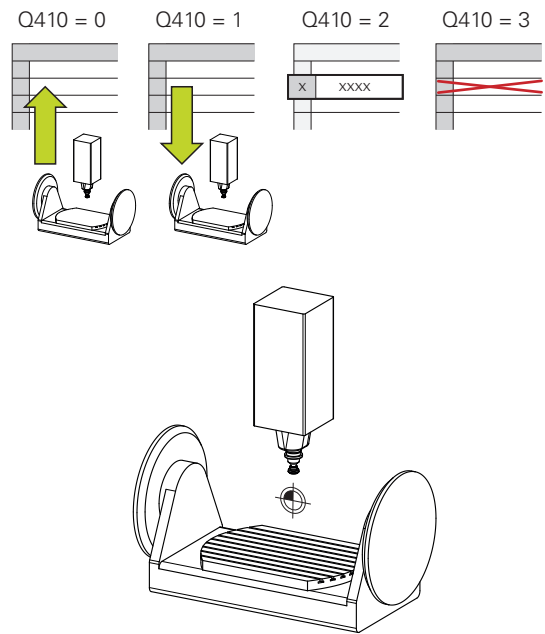
With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.

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.

ℹ️ 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** does not 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 are 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 (**TCHPRAUTO.html**) 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

35.3.2   Cycle 451 MEASURE KINEMATICS (#48 / #2-01-1)

ISO programming

G451

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
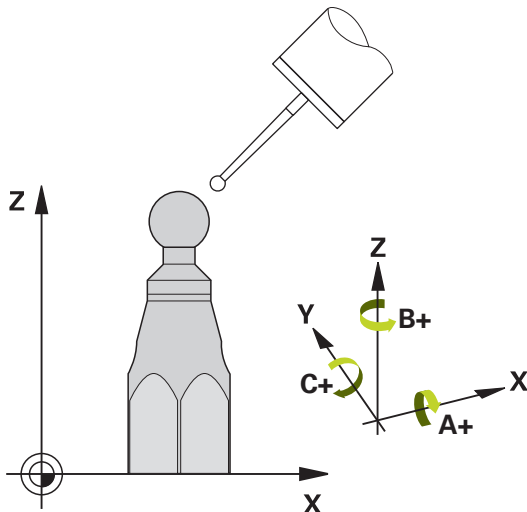
Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

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 run

1 Clamp the calibration sphere and check for potential collisions.

2 In the **Manual operation** operating 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.

Result parameter Q

The control saves the results of the touch probe cycle 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

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 - QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"**

Q parameter number	Meaning
QS144	Position error of A axis
	E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}
QS145	Position error of B axis
	E_{Z0B} E_{X0B} E_{C0B} E_{A0B}
QS146	Position error of C axis
	E_{X0C} E_{Y0C} E_{A0C} E_{B0C}

> **i** Position faults are deviations from the ideal axis position and are marked by four characters.
>
> Example: E_{X0C} = Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)	; Read out the first result E_{X0C} from **QS146**
12 QL0 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL0**
13 QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)	; Read out the second result E_{Y0C} from **QS146**
14 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL1**
15 QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)	; Read out the third result E_{A0C} from **QS146**
16 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL2**
17 QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)	; Read out the forth result E_{B0C} from **QS146**
18 QL3 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL3**

Further information: "String functions", Page 1048

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). There is a danger of collision!
>
> ▶ So remember to leave a large enough set-up clearance to prevent any risk of collision between the touch probe and calibration sphere
>
> ▶ Also ensure that there is enough space to reach the set-up 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 the machine parameter **mStrobeRotAxPos** (no. 204803) for this purpose. There is a danger of collision!
>
> ▶ Note the documentation of the machine manufacturer

> **ⓘ** ■ Define the retracting height above 0 if software option (#9 / #4-01-1) 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), for example 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.

> **i** 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 calibration sphere 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.

> **i** 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 output by the control in the measurement 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.

> ℹ If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

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.

> ℹ 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.
>   **Further information:** "Log function", Page 1598

Notes

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D 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 3D 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.

■ After measuring the kinematics, you must re-determine the preset.

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**. KinematicsComp (#52 / #2-04-1) is the prerequisite for angle error compensation.
	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**. KinematicsComp (#52 / #2-04-1) is the prerequisite.
	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 active 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**. This 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. This 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. This 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. This 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. This 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. This 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. This value has an absolute effect.
	Input: **-359.9999...+359.9999**
	Q420 End angle in C axis?

Help graphic	Parameter
	End angle in the C axis at which the last measurement will be made. This 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**
	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.

Help graphic	Parameter
	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.

"Optimize position and angle" mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> **i** 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.
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> **i**
> - For correct determination of the angular position errors, HEIDENHAIN recommends setting the affected rotary axis to an inclination angle of 0° for this measurement.
> - After correcting a machine datum, the control tries to reduce the compensation of the associated angular position error (**locErrA**/**locErrB**/**locErrC**) of the measured rotary axis.

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 creates a log (**TCHPRAUTO.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
- Tool name
- Active kinematics
- Mode used (0=Check/1=Optimize position/2=Optimize pose/3=Optimize machine datum and pose)
- Inclination angles
- For each measured rotary axis:
    - Starting angle
    - End angle
    - Number of measuring points
    - Measuring circle radius
    - Averaged backlash, if **Q423>0**
    - Positions of the axes
    - Standard deviation (scatter)
    - Maximum deviation
    - Angular error
    - 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)
    - Averaged positioning error and standard deviation of the positioning errors to 0
    - SVG files with graphs: measured and optimized errors of individual measurement positions.
        - Red curve: measured positions
        - Green curve: optimized values after cycle has run
        - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = component error in Y of axis C)
        - X axis of the graph: rotary axis position in degrees
        - Y axis of the graph: position deviations in mm
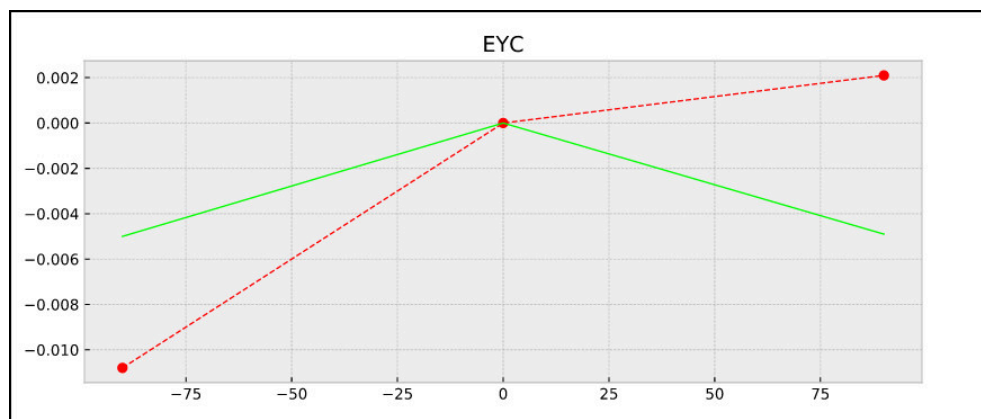
Sample measurement: EYC component error in Y of axis C

35.3.3 Cycle 452 PRESET COMPENSATION (#48 / #2-01-1)

ISO programming

G452

Application

> ⚙ Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
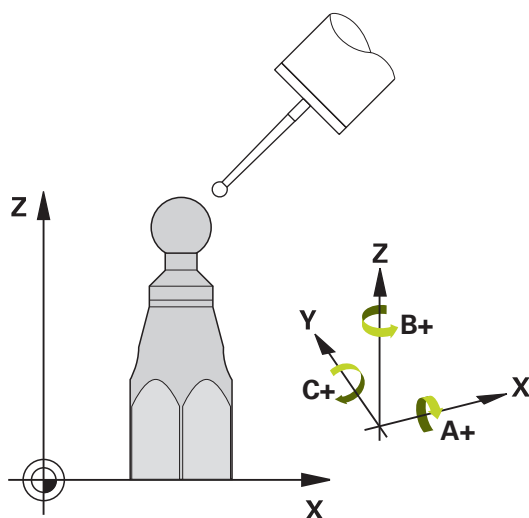
Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (#48 / #2-01-1)", Page 1583). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is at the center of the calibration sphere after optimization.

Cycle run

> **i** 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.

Result parameter Q

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

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 - QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"**

Q parameter number	Meaning
QS144	Position error of A axis
	E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}
QS145	Position error of B axis
	E_{Z0B} E_{X0B} E_{C0B} E_{A0B}
QS146	Position error of C axis
	E_{X0C} E_{Y0C} E_{A0C} E_{B0C}

> **i** Position faults are deviations from the ideal axis position and are marked by four characters.
>
> Example: E_{X0C}= Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)	; Read out the first result E_{X0C} from **QS146**
12 QL0 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL0**
13 QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)	; Read out the second result E_{Y0C} from **QS146**
14 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL1**
15 QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)	; Read out the third result E_{A0C} from **QS146**
16 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL2**
17 QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)	; Read out the forth result E_{B0C} from **QS146**
18 QL3 = TONUMB (SRC_QS0)	; Convert alphanumeric value from **QS0** to a numerical value and assign it to **QL3**

Further information: "String functions", Page 1048

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 3D 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 3D 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 active 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**. This 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. This 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. This 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. This 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)?

Help graphic	Parameter
	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. This 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. This 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. This 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. This 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**
	Q432 Angular range of backlash comp.?

Help graphic	Parameter
	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 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

> ℹ️ 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 (**TCHPRAUTO.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
- Tool name
- Active kinematics
- Mode used
- Inclination angles
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Number of measuring points
  - Measuring circle radius
  - Averaged backlash, if **Q423>0**
  - Positions of the axes
  - Standard deviation (scatter)
  - Maximum deviation
  - Angular error
  - Compensation values in all axes (preset shift)
  - 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)
  - Averaged positioning error
  - SVG files with graphs: measured and optimized errors of individual measurement positions.
    - Red curve: measured positions
    - Green curve: optimized values
    - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = deviations) of the Y axis in dependency of the C axis.
    - X axis of the graph: rotary axis position in degrees
    - Y axis of the graph: position deviations in mm
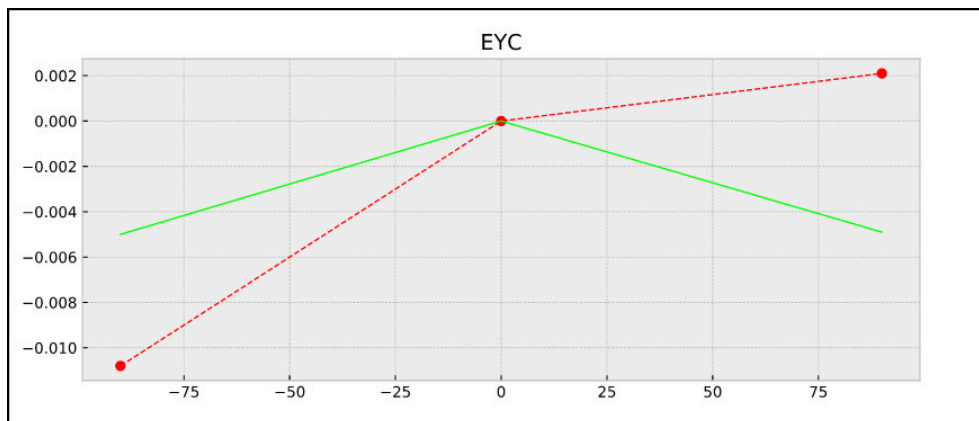
Sample measurement: EYC deviations of the Y axis in dependency of the C axis

35.3.4 Cycle 453 KINEMATICS GRID (#48 / #2-01-1)

ISO programming
G453

Application

> ⚙ Refer to your machine manual.
>
> The KinematicsOpt software option (#48 / #2-01-1) is required.
>
> This function must be enabled and adapted by the machine manufacturer.
>
> To use this cycle, your machine manufacturer needs to create and configure a compensation table (*.kco) first and enter some more settings.
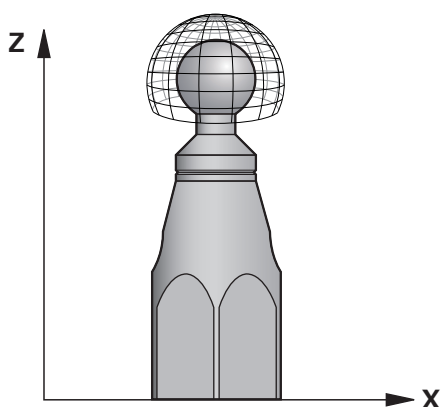
Even if your machine was already optimized regarding positioning errors (e.g., via Cycle **451**), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. These can result, for example, from component errors (e.g., a bearing error) with head rotation axes.

Cycle **453 KINEMATICS GRID** enables errors in swivel heads to be detected and compensated for in accordance with the rotary axis positions. With this cycle and using a 3D 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 manufacturer defines these swivel axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing operation on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine manufacturer 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 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. Use, for example, Cycle **451** before doing so.

> ℹ HEIDENHAIN recommends using the calibration spheres **KKH 250 (order number 655475-01)** or **KKH 100 (order 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 run

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)

> **i**   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)

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

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.

> **i**   Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes

⚙️ The software option (#48 / #2-01-1) is required.

This function must be enabled and adapted by the machine manufacturer.

Your machine manufacturer defines the storage location of the compensation table (*.kco).

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D 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 3D 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 amount of the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.

ℹ️
- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D 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). (#52 / #2-04-1)
	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). (#52 / #2-04-1)
	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 active 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**. This 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?

Help graphic	Parameter
	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. This 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 creates a log (**TCHPRAUTO.html**) and saves it in the folder where the current NC program resides. It contains the following data:

- Date and time of log 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

36

Pallet Machining and Job Lists

36.1 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), fixtures (FIX) optionally, 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 1630

36.1.1 Pallet counter

You can define a pallet counter on the control. This allows you to define the number of parts produced variably (e.g., in case of pallet handling with automatic workpiece change).

To do this, define a nominal value in the **TARGET** column of the pallet table. The control repeats the NC programs of this pallet until the nominal value is reached.

By default, every processed NC program raises the actual value by 1. If, for example, an NC program produces several workpieces, define the value in the **COUNT** column of the pallet table.

Further information: "Pallet table *.p", Page 1727

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 1621

36.2 The Job list workspace

36.2.1 Fundamentals

Application

In the **Job list** workspace, you edit and execute pallet tables.

Related topics

- Contents of a pallet table
  Further information: "Pallet table *.p", Page 1727
- The **Form** workspace for pallets
  Further information: "The Form workspace for pallets", Page 1628
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1630

Requirement

■ Batch Process Manager software option (#154 / #2-05-1)

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.

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 1621

If you activate the **Edit** toggle switch, the **Insert row** button will be displayed in the action bar and allows you to insert a new table row.

Further information: "The Insert row window", Page 1623

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 1622
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 1620
	Machining method:
	■ Workpiece-oriented machining
	■ Tool-oriented machining
	Further information: "Machining method", Page 1622
Sts	Machining status
	Further information: "Machining status", Page 1622

Status of the pallet, fixture, or NC program

The control uses the following icons to display the status:

Icon	Meaning
	Pallet, **Fixture** or **Program** is locked
	Pallet or **Fixture** is not enabled for machining
	This line is currently being executed in **Program run, single block** or **Program run, full sequence** operating 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 symbol	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

The **Insert row** window
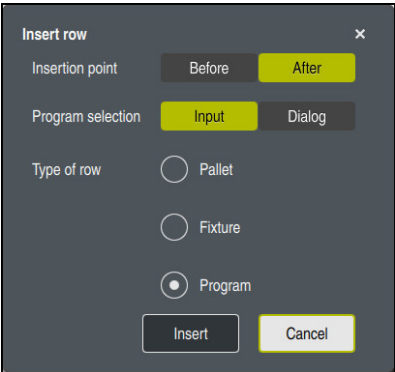
The **Insert row window** with the **Program** selection

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 row	Corresponds to the **TYPE** column of the pallet table Insert a **Pallet**, **Fixture** or **Program**

You can edit the contents and settings of a row in the **Form** workspace.

Further information: "The Form workspace for pallets", Page 1628

The 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 1624

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 **editTableWhileRun** (no. 202102), the machine manufacturer defines whether you will be allowed to edit the pallet table during program run.
- 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 optional machine parameter **resumePallet** (no. 200603), the machine manufacturer defines whether the control will continue program execution after an error message.
- The optional machine parameter **failedCheckReact** (no. 202106) allows you to define whether the control checks incorrect tool or program calls.
- The optional machine parameter **failedCheckImpact** (no. 202107) allows you to define whether the control skips the NC program, the fixture or the pallet after an incorrect tool or program call.

36.2.2 Batch Process Manager (#154 / #2-05-1)

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

- The **Job list** workspace
  Further information: "The Job list workspace", Page 1620
- Editing a pallet table in the **Form** workspace
  Further information: "The Form workspace for pallets", Page 1628
- Contents of the pallet table
  Further information: "Pallet table *.p", Page 1727

Requirements

- Batch Process Manager software option (#154 / #2-05-1)

  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 1782

Description of function
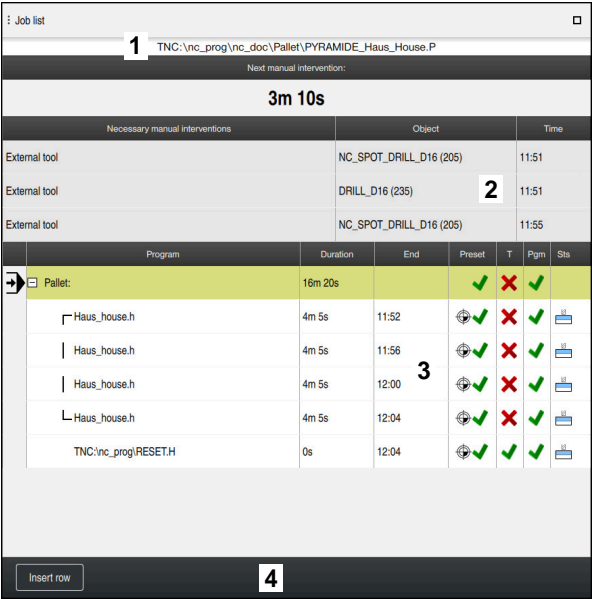
The **Job list** workspace with **Batch Process Manager** (#154 / #2-05-1)

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 1627

4 Action bar

If the **Edit** toggle switch is active, you can add a new row.

If the **Edit** toggle switch is inactive, you can use Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) to check all NC programs of the pallet table in the **Program Run** operating mode.

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 1622
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 1620 Machining method: ▪ Workpiece-oriented machining ▪ Tool-oriented machining **Further information:** "Machining method", Page 1622
Duration	Duration of executing the pallet, fixture, or NC program
End	Expected point in time after execution of the NC program In the **Editor** operating mode, the **End** column does not show a point of time but the duration.
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 1628
T	Status of the tools used: ▪ Test completed ▪ Test not yet completed ▪ Test failed The column only shows the status in the **Program Run** operating mode. **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1628
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 1628
Sts	Machining status **Further information:** "Machining status", Page 1622

Status of the workpiece preset, the tools, and the NC program

The control uses the following icons to display the status:

Icon	Meaning
✔	Test completed
✔	Collision checking completed Program simulation with active **Dynamic Collision Monitoring (DCM)** (#40 / #5-03-1)
�’	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 ✔.

36.3 The Form workspace for pallets

Application

In the **Form** workspace the control shows the contents of the pallet table for the selected row.

Related topics

- The **Job list** workspace
  Further information: "The Job list workspace", Page 1620
- Contents of the pallet table
  Further information: "Pallet table *.p", Page 1727
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1630

Description of function

The **Form** workspace with the contents of a pallet table

A pallet table can have the following types of rows:

- **Pallet**
- **Fixture**
- **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 the **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 toggle switches provided in the form correspond to the following table columns:

- The **Locked** toggle switch corresponds to the column **LOCK**
- The **Machinable** toggle 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.

36.4 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 *.p", Page 1727
- Block scan for mid-program startup in a pallet table
  Further information: "Block scan in pallet tables", Page 1655

Requirements

- 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** (#9 / #4-01-1)
    Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787
  - **M144** (#9 / #4-01-1)
    Further information: "Taking the tool offset into account in calculations M144 (#9 / #4-01-1)", Page 995
  - **M101**
    Further information: "Automatically inserting a replacement tool with M101", Page 999
  - **M118** (#21 / #4-02-1)
    Further information: "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979
  - Changing the pallet preset
    Further information: "Pallet preset table", Page 1635

Description of function

The following columns of the pallet table apply to 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 manufacturer processes them in the NC macros.

Further information: "Pallet table *.p", Page 1727

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 1174

Sequence of tool-oriented machining

1 The entries TO and CTO tell the control that tool-oriented machining is in effect 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

Mid-program startup with block scan

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.

If you use the block scan to start in a pallet table, the control will always execute the chosen row in the pallet table as a workpiece-oriented process.

After a block scan, the control can resume tool-oriented machining if the tool-oriented machining method TO and CTO is defined in the subsequent rows.

Further information: "Pallet table *.p", Page 1727

⚙️ Refer to your machine manual.

Tool-oriented machining is a machine-dependent function. The standard functional range is described below.

Tool-oriented machining allows you to machine several workpieces together even on a machine without pallet changer, which reduces tool-change times.

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

---

The following functions are not permitted:
■ FUNCTION TCPM, M128
■ M144
■ M101
■ M118
■ Changing the pallet preset

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**
■ Tilting the working plane

Unless the machine manufacturer has made a different configuration, you need the following additional columns for tool-oriented machining:

Column	Meaning
W-STATUS	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
METHOD	Indicates the machining method
	Tool-oriented machining is also possible with a combination of pallet fixtures, but not for multiple pallets.
	The control differentiates between the following entries
	■ WPO: Workpiece oriented (standard)
	■ TO: Tool oriented (first workpiece)
	■ CTO: Tool oriented (further workpieces)
CTID	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.
SP-X, SP-Y, SP-Z, SP-A, SP-B, SP-C, SP-U, SP-V, SP-W	The entry for the clearance height in the existing axes is optional.
	You can enter safety positions for the axes. The control only approaches these positions if the machine manufacturer processes them in the NC macros.

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 in effect 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-program 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.

36.5 Pallet preset table

Application

Pallet presets are an easy way to compensate, for example, for mechanical differences between individual pallets.

The machine manufacturer defines the pallet preset table.

Related topics

- Contents of the pallet table

  Further information: "Pallet table *.p", Page 1727

- Workpiece preset management

  Further information: "Preset management", Page 705

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, for example, positioning the pallet preset in the center of a clamping tower.

When a pallet preset is active, the control displays an icon with the number of the active pallet preset in the **Positions** workspace.

Further information: "The Positions workspace", Page 149

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 (#17 / #1-05-1)", Page 1257

Notes

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. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

▶ Refer to the machine manufacturer's documentation
▶ Use pallet presets only in conjunction with pallets
▶ Change pallet presets only after discussion with the machine manufacturer
▶ Check the pallet preset in the **Setup** application before you start machining

---

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 pallet preset in the **Setup** application before you start machining
▶ 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 708

37

Program Run

37.1 The Program Run operating mode

37.1.1 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: "The MDI Application ", Page 1221
- Creating NC programs

  Further information: "Programming fundamentals", Page 199
- Pallet tables

  Further information: "Pallet Machining and Job Lists", Page 1619

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

Danger of collision!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. Undesirable machine movements or collisions may result.

▶ Edit NC programs in the **Program** workspace only

Description of function

> **i** 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 1649

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 **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 TNC bar", Page 155

The **Program Run** operating mode provides the following workspaces:

- **Positions**

  Further information: "The Positions workspace", Page 149

- **Program**

  Further information: "The Program workspace", Page 203

- **Simulation**

  Further information: "The Simulation Workspace", Page 1197

- **Status**

  Further information: "The Status workspace", Page 157

When opening a pallet table, the control displays the **Job list** workspace. You cannot modify this workspace.

Further information: "The Job list workspace", Page 1620

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
📄	**Open file** **Open file** allows you to open a file, such as an NC program. When 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 executed or is marked for execution.
Single Block	If this toggle 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:** "The Q parameter list window", Page 1010
Compensation tables	The control opens a selection menu with the following tables: ▪ **D** ▪ **T-CS** ▪ **WPL-CS** **Further information:** "Compensation during program run", Page 1658
GOTO Cursor	The control marks the table row currently selected for execution. Active only if a pallet table is open (option 22) **Further information:** "The Job list workspace", Page 1620
F limited	Use this option to activate or deactivate the feed-rate limit for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 1774
AFC	Use this option to activate or deactivate Adaptive Feed Control (AFC, option 45). **Further information:** "The AFC toggle switch in the Program Run operating mode", Page 889
AFC settings	The control opens a selection menu with the following tables for AFC (option 45): ▪ **AFC.TAB** for AFC basic settings ▪ **AFC.DEP** settings file for teach-in cuts of the active NC program ▪ **AFC2.DEP** log file of the active NC program **Further information:** "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 893
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 1642

Icon or button	Meaning
Breakpoints	If you select this button, the control opens the **Breakpoints** window with the following options for selection:
	■ **Permit start with override**
	If the toggle switch is active, then you can continue the NC program after a conditional stop with the Override Controller:
	Further information: "The Program run options window", Page 1759
	■ **Feed F LIMIT**
	Use this function to activate a feed-rate limit and define its value.
	Further information: "Feed rate limit F LIMIT", Page 1642
	■ **Perform conditional stop**
	The control provides the following breakpoints:
	■ **Before switch to rapid traverse**
	■ **Before switch to feed rate**
	■ **Between two rapid traverses**
	■ **Before tool call**
	■ **Before tilting the working plane**
	■ **Before cycle call**
	■ **In cycle call**
	Further information: "Breakpoints", Page 1761
	■ **Skip block**
	If the toggle switch is active, then the control does not execute any NC blocks dimmed with the **/** character.
	Further information: "Hiding NC blocks", Page 1165
	If the toggle switch is active, then the dims the NC blocks to be skipped.
	Further information: "Appearance of the NC program", Page 205
	■ **Pause at M1**
	If the toggle switch is active, then the control stops the execution at every NC block with **M1**.
	Further information: "Overview of miscellaneous functions", Page 965
	If the toggle switch is inactive, then the control dims the **M1** syntax element.
	Further information: "Appearance of the NC program", Page 205
Skip block	If the toggle switch is active, then the control does not execute any NC blocks dimmed with the **/** character.
	Further information: "Hiding NC blocks", Page 1165
	If the toggle switch is active, then the dims the NC blocks to be skipped.
	Further information: "Appearance of the NC program", Page 205
Pause at M1	If the toggle switch is active, then the control stops the execution at every NC block with **M1**.
	Further information: "Overview of miscellaneous functions", Page 965
	If the toggle switch is inactive, then the control dims the **M1** syntax element.
	Further information: "Appearance of the NC program", Page 205
GOTO block number	Mark an NC block to be run without considering any previous NC blocks
	Further information: "GOTO function", Page 1163
Manual traverse	While a program run is interrupted, you can move the axes manually.

Icon or button	Meaning
	If **Manual traverse** is active, then the operating mode's icon in the control bar changes.
	Further information: "Manual traverse during an interruption", Page 1648
Edit	If this toggle switch is active, then you can edit the pallet table.
	Active only if a pallet table is open
	Further information: "The Job list workspace", Page 1620
3D ROT	While a program run is interrupted, you can move the axes manually in the tilted working plane (option 8).
	Further information: "Manual traverse during an interruption", Page 1648
Approach position	Return to contour after manual traverse of the machine axes during an interruption
	Further information: "Returning to the contour", Page 1657
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 1649
Open in the editor	The control opens the active NC program in the **Editor** operating mode, even if it is a called NC program.
	Active only if an NC program is open
	Further information: "The Editor operating mode", Page 202
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 active program information as well as the program run-time.

Feed rate limit F LIMIT

The **F LIMIT** 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 **F LIMIT** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **F LIMIT** button in the function bar, the control will open the **Feed rate F LIMIT** window.

If a feed rate limit is active, the control highlights the **F LIMIT** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the feed rate is displayed in orange.

Further information: "Statusanzeigen", Page

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate F LIMIT** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt 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 and the **Internal stop** button)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 1193

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 TNC bar", Page 155

Below are some of the functions you can execute in an interrupted or canceled state:

- Selecting an operating mode
- Manual traverse of axes
- Checking Q parameters and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the 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 **Q parameter list** window
  - 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**

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 1649

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 1657

> **ℹ** The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE
Danger of collision!
Program cancellation, manual intervention, 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 **Single Block** mode

- 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 positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece preset during a program run interruption, you must re-select the NC block to resume.
  Further information: "Block scan for mid-program startup", Page 1649
- 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 execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

Definitions

Abbreviation	Definition
GPS (global program settings)	Global program settings
ACC (active chatter control)	Active Chatter Control

37.1.2   Navigation path in the Program workspace

Application

If you execute an NC program or a pallet table or if you test it in the opened **Simulation** workspace, the control will display a navigation path in the file information bar of the **Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

Related topics

- Program call

  Further information: "Selection functions", Page 364

- **Program** workspace

  Further information: "The Program workspace", Page 203

- **Simulation** workspace

  Further information: "The Simulation Workspace", Page 1197

- Interrupted program run

  Further information: "Interrupting, stopping or canceling program run", Page 1643

Requirement

- The **Program** and **Simulation** workspaces are both opened

  In the **Editor** operating mode you need both workspaces to use the function.

Description of function

The control shows the name of the NC program as a path element in the file information bar. As soon as the control calls a different NC program, the control adds a new path element with the name of the called NC program to the bar.

Additionally, the control displays the contents of the called NC program in a new pane in the **Program** workspace. The control displays as many NC programs side by side as the size of the workspace permits. If necessary, newly opened NC programs will cover previously opened NC programs. The control displays the covered NC programs in a narrow band at the left edge of the workspace.

When execution is interrupted, you can navigate between the NC programs. When you select the path element of an NC program, the control opens the content.

When you select the last path element, the control automatically marks the active NC block with the execution cursor. When you press the **NC Start** key, the control resumes execution of the NC program from this position.
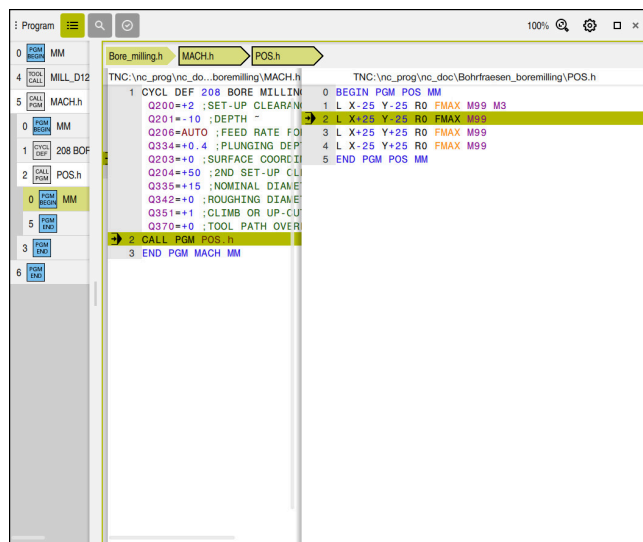
Called NC programs in the **Program** workspace in the **Program Run** operating mode

Depiction of path elements

The control displays the path elements of the navigation path as follows:

Format	Meaning
Black frame	The NC program is visible in the **Program** workspace and is not covered by other NC programs.
Highlighted in green	The NC program at the current cursor position is active or is considered for program run. If, for example, the cursor is positioned in the called NC program, the calling NC program will be considered for program run.
Highlighted in gray	The NC program is active for execution but will not be considered for program run at the current cursor position. If, for example, you stop the execution and navigate into the calling NC program, the control displays the path element of the called NC program in gray.

Note

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.

The structure items allow you to navigate into every NC program. The control displays the associated NC programs in the **Program** workspace. The navigation path always remains at the current point of execution.

Further information: "The Structure column in the Program workspace", Page 1166

37.1.3 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

The **Tilt the working plane (3D ROT)** window allows selecting the reference system in which you move the axes (#8 / #1-01-1).

Related topics

■ Manual traverse of machine axes

Further information: "Moving the machine axes", Page 188

■ Tilting the working plane manually (#8 / #1-01-1)

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 736

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 189

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 692
⬚	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS**
		Further information: "Workpiece coordinate system W-CS", Page 697
⬚	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS**
		Further information: "Working plane coordinate system WPL-CS", Page 699
⬚	**T-CS tool**	Traversing in the tool coordinate system **T-CS**
		Further information: "Working plane coordinate system WPL-CS", Page 699

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). Selecting an incorrect **3D ROT** setting or moving the tool in the wrong direction involves risk of collision!
▶ It is better to use the **T-CS** function
▶ Check the direction of movement
▶ Move at slow 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.

37.1.4 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 199
- Pallet tables and job lists

  Further information: "Pallet Machining and Job Lists", Page 1619

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:

- The **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 1652
- Multi-level block scan in subprograms and touch probe cycles
  Further information: "Performing a multi-level block scan", Page 1653
- Block scan in a point table
  Further information: "Block scan in point tables", Page 1654
- Block scan in pallet programs
  Further information: "Block scan in pallet tables", Page 1655

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During the block scan you can activate or deactivate **Single Block** mode.

The Block scan window

The **Block scan** window with saved interruption point and open **Point table** area

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

Block scan
▶ 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

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 displays 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 displays 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.

> **ℹ** You can also position the axes individually in a self-selected sequence.
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1658

▶ 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

▶ 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 1652

▶ Activate the **Single Block** toggle switch as needed

▶ Press the **NC Start** key to execute individual NC blocks as needed

▶ 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 displays 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 displays 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 1658

▶ 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 displays 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 displays 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.

ℹ You can also position the axes individually in a self-selected sequence.
Further information: "Positioning the axes in a self-selected sequence", Page 1658

ℹ 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: "Pattern definition cycles", Page 405

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 displays 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 displays 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.

> **ℹ** You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1658

> **ℹ** 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 in multiple steps

■ The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. When the control reads such a NC block during the **Block scan**, it will also display an error message.

■ The control only displays the dialogs required by the process in the pop-up window.

■ If you use the block scan to start in a pallet table, the control will always execute the chosen row in the pallet table as a workpiece-oriented process. After the pallet table line selected in the **Block scan**, the control resumes machining according to the defined machining method.

Further information: "Tool-oriented machining", Page 1630

■ 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 161

■ 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.

37.1.5 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 1648
- The **Block scan** function
  Further information: "Block scan for mid-program startup", Page 1649

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.

The **Axis sequence for return to contour: window**

The **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 1658

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 1658

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

	▶	Select **Approach position**
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**)
[NC Start icon]	▶	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 check mark for **Target**.
	▶	Position the remaining axes
	>	When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

	▶	Select **Approach position**
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.

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

37.2 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 801
- Editing compensation tables in the NC program
  Further information: "Accessing table values ", Page 1679
- Contents and creation of compensation tables
  Further information: "Compensation table *.tco", Page 1732
  Further information: "Compensation table *.wco", Page 1734
- Contents and creation of a datum table
  Further information: "Datum table", Page 715
- Activating a datum table in the NC program
  Further information: "Datum table *.d", Page 1721

Description of function

The control opens the selected tables in the **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

37.2.1 Opening tables from within the Program Run operating mode

To open the compensation tables from within the **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 the **Tables** operating mode.

Notes

<table>
<tr><td align="center">***NOTICE***</td></tr>
<tr><td>

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

</td></tr>
</table>

■ When opening a table in the **Program Run** operating mode, the control will display the **M** status in the table tab. This status means that this table is active for the program run.

■ The clipboard allows you to transfer axis positions from the position display to the datum table.

Further information: "Status overview on the TNC bar", Page 155

37.3 The 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** toggle switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**

  Further information: "The Retract workspace", Page 1662
- **Positions**

  Further information: "The Positions workspace", Page 149
- **Status**

  Further information: "The Status workspace", Page 157

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** toggle switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- The **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 LIMIT", Page 1642

The Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction:
	▪ **Machine axes**: Move in the machine coordinate system **M-CS**
	▪ **Tilted system**: Move in the working plane coordinate system **WPL-CS** (#8 / #1-01-1)
	▪ **Tool axis**: Move in the working plane coordinate system **T-CS** (#8 / #1-01-1)
	▪ **Thread**: Move in the tool coordinate system **T-CS** with compensating movements of the spindle
	Further information: "Reference systems", Page 690
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes
	Effective in the **Tilted system** traverse mode
Thread pitch	Thread pitch from the **PITCH** column of tool management
	Effective in the **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool:
	▪ **Right-hand thread**
	▪ **Left-hand thread**
	Effective in the **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect
	Effective in the **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 in the tilted working plane was being performed. 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
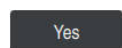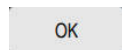
Retract ▶ Activate the **Retract** toggle switch

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

> The control opens the **Retract** application and displays the **Assume position values?** window.

▶ Compare the displayed position values with the actual position values

OK ▶ 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

Retract ▶ Select **Retract**

▶ Retract the tool with the axis keys or the handwheel

End retraction ▶ Select **End retraction**

> The control opens the **End retraction?** window and prompts you to answer a confirmation request.

Yes ▶ If the tool was correctly retracted, select **Yes**

> The control closes the **End retraction?** window and the **Retract** application.

38

Tables

38.1   The 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 displays the **Quick selection new table** and **Open File** workspaces.

In the **Quick selection new table** workspace you can create a new table and open some tables directly.

Further information: "Quick selection workspaces", Page 835

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "The Open File workspace", Page 834

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. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table** and **Form** workspaces in every application.

Further information: "The Table workspace", Page 1670

Further information: "The Form workspace for tables", Page 1676

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 1174

Buttons

In the **Tables** operating mode, the function bar contains the following buttons that can be used for any table:

Button	Meaning
Undo	The control undoes the last change.
Redo	The control restores the change that was undone.
GOTO record	The control opens the **GOTO jump instruction** window. The control jumps to the row number you have defined.
Edit	If the toggle switch is active, you can edit the table.
Reset row	The control resets all data contained in the row.
Mark row	The control marks the currently selected row.

Depending on the selected table, the control provides the following additional buttons in the function bar:

Button	Meaning
Insert rows	The control opens the **Insert rows** window where you can insert one or more new rows. If you enable the **Append** checkbox, the control will insert the rows after the last table row.
Delete rows	The control deletes the currently selected row.
Insert tool	The control opens the **Insert tool** window where you can define the following: ■ **Type:** **Further information:** "Tool types", Page 261 ■ **Line number (Tool number?)** ■ **Number of rows** ■ **Index** **Further information:** "Indexed tool", Page 258 ■ **Append** Append rows at the end of the table **Further information:** "Tool management ", Page 269
Delete tool	The control deletes the tool selected in the tool management. You cannot delete any tools that have been entered into the pocket table. The button is dimmed. **Further information:** "Tool management ", Page 269
Import	The control imports tool data. **Further information:** "Importing tool data", Page 271
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.
Activate the preset	The control activates the currently selected row of the preset table as preset. **Further information:** "Preset table *.pr", Page 1710
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes. **Further information:** "Write-protection for table rows", Page 1715

⚙ Refer to your machine manual.
If necessary, the machine manufacturer adapts the buttons.

38.1.1 Editing the contents of tables

To edit the contents of a table:

▶ Select the desired table cell

Edit
⬤━
    ▶ Enable **Editing**
    > The control enables the values for editing.

ℹ To edit the contents of a table, you can also double-tap or double-click the table cell. The control displays the **Editing disabled. Enable?** window. You can enable the values for editing or abort the process.

ℹ If the **Editing** toggle switch is enabled, you can edit the contents both in the **Table** workspace and in the **Form** workspace.

Notes

- The control enables you to transfer tables from previous controls to the TNC7 basic and to adapt them automatically, if needed.

- When you open a table where columns are missing, for example in case of a tool table from a previous control, the control will display the **Incomplete table layout** window.

  When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

  In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.

- If you, for example, have processed tables in a text editor, the control offers the **Update TAB / PGM** function. Use this function to complete an incorrect table format.
  Further information: "File management", Page 822

  ℹ Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., format errors).

- Refer to your machine manual.
  Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. This machine parameter allows to define columns as required fields or to reset them automatically to a default value. If a rule is violated, the control displays a note icon.

38.2 The Create new table window

Application

You can create tables using the **Create new table** window in the **Quick selection new table** workspace.

Related topics

- The **Quick selection new table** workspace

  Further information: "Quick selection workspaces", Page 835
- Available file types for tables

  Further information: "File types", Page 829

Description of function
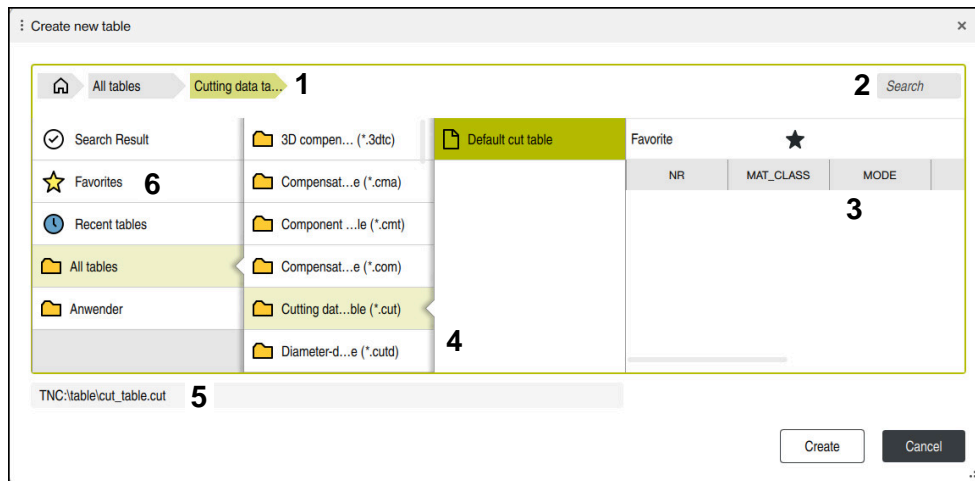
The **Create new table** window

The **Create new table** window shows the following areas:

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. Searching

   You can search for any strings. The control displays the results under **Search Result**.

3. The control shows the following information and functions:
   - Add or remove a favorite
   - Preview

4. Content columns

   The control shows a folder and the available prototypes for each table type.

5. Path of the table to be created

6. Navigation column

   The navigation column contains the following areas:
   - **Search Result**
   - **Favorites**

     The control displays all folders and prototypes that you have marked as favorites.
   - **Last functions**

     The control shows the eleven most recently used prototypes.
   - **All functions**

     The control shows all available table types in the folder structure.

Notes

- 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.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer can provide additional areas in the navigation column (e.g., tables for the user).
- With the optional machine parameter **dialogText** (no. 105506), the machine manufacturer can define other names for the table types (e.g., tool table instead of **t**).

38.3 The 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

The **Table** workspace

In the 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.

> ⚙ Refer to your machine manual.
> If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Icons and shortcuts

The **Table** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
☰	Open or close the **Filter** column **Further information:** "The Filter column in the Table workspace", Page 1671
🔍 **CTRL + F**	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 1674
‹|›	Enable or disable **Change column width**
🖉	**Edit table characteristics** **Further information:** "Modifying the properties of freely definable tables", Page 1709
100%	Current size of the content Open or close the **Scaling** selection menu
🔍	**Reset scaling** Set the font size of the table to 100%
⚙	Open or close settings in the **Tables** window **Further information:** "Settings in the Table workspace", Page 1674
CTRL + A	Mark all rows
CTRL + SPACE	Mark the active row or end the marking function
SHIFT + UP	Additionally mark the row above
SHIFT + DOWN	Additionally mark the row below

The Filter column in the Table workspace

You can filter the following table types:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

When you tap or click a filter once, the control activates the selected filter in addition to the currently active filters. When you double-tap or double-click a filter, the control activates only the selected filter and deactivates all other filters.

Filtering in the Tool management

The control provides the following default filters in the **Tool management**:

- **All tools**
- **Magazine tools**

According to the selection of **All tools** or **Magazine tools**, the control additionally offers the following default filters in the filter column:

- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Touch probes** (#17 / #1-05-1)
- **Undefined tools**

Filtering in the Pocket table

The control provides the following default filters in the **Pocket table**:

- **all pockets**
- **spindle**
- **main magazine**
- **empty pockets**
- **occupied pockets**

Filtering in the Presets table

The control provides the following default filters in the **Presets** table:

- **Base transformations**
- **Offsets**
- **SHOW ALL**

User-defined filters

You can additionally create user-defined filters.

The control provides the following icons for each user-defined filter:

Icon	Meaning
✏	If you click **Edit**, the control opens the **Search** column. You can edit and save the selected filter or save a filter under a new name. **Further information:** "The Search column in the Table workspace", Page 1674
🗑	You can delete the selected filter.

If you want to deactivate the user-defined filters, you have to double-tap or double-click the **All** filter.

> ⚙ 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.

Logical connective operations between requirements and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter

  You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.

- OR operation between filters of the same type

  When you activate the default filters **Milling cutters** and **Lathe tools**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the requirements. The table row must contain either a milling cutter or a turning tool.

- AND operation between filters of different types

  You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

The Search column in the Table workspace

You can search the following table types:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

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 if applicable (e.g., **Contains** or **Equal to (=)**)

  Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

> **i** If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use **Add** to add several conditions. The conditions will have a combined effect when you perform the search.
	You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a filter. You can assign any name to the filter.

> **⚙** 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.

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**

The General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** toggle switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control moves the cursor synchronously in the **Form** workspace.

The Column sequence area
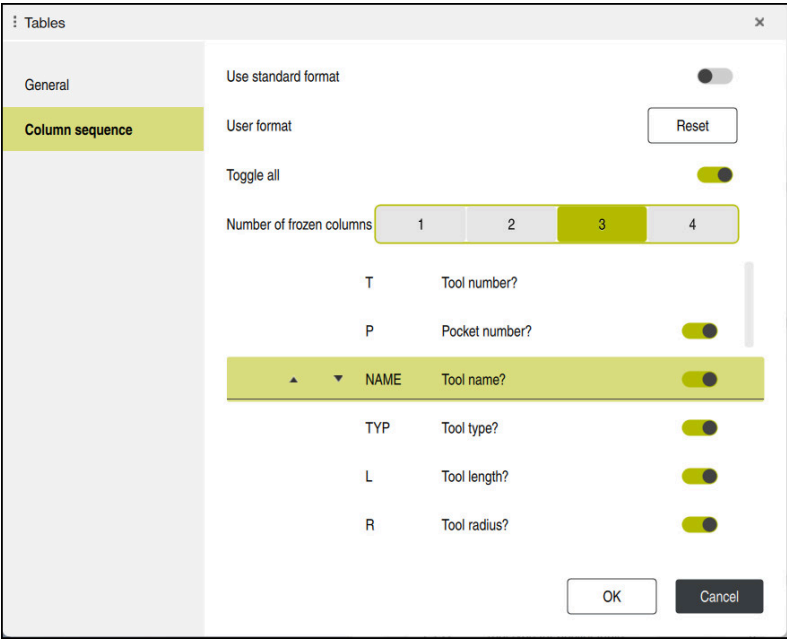
The **Tables** window

The **Column sequence** area contains the following settings:

Setting	Meaning
Use standard format	If you activate the toggle switch, the control shows all table columns, indicating them in the standard sequence.
	If you deactivate the toggle switch, the control restores the previous setting.
User format	If you select the **Reset** button, the control resets the adaptations to the settings of the standard format.
Toggle all	If you activate the toggle switch, the control shows all table columns.
	If you deactivate the toggle switch, the control hides all table columns.
	The first column in each table cannot be hidden.
Number of frozen columns	You define how many table columns the control freezes at the left table edge. You can freeze up to four table columns.
	These table columns will remain visible even when you navigate further to the right within the table.
Columns of the currently opened table	The control displays all table columns below each other. Use the toggle switches to separately hide or show each table column.
	The control displays a line below the selected number of frozen columns.
	When you select a table column, the control displays up and down arrows. Use these arrows to change the sequence of the columns.
	The respective first column in the table cannot be shifted.

The settings in the **Column sequence** area only apply to the currently opened table.

38.4 The 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
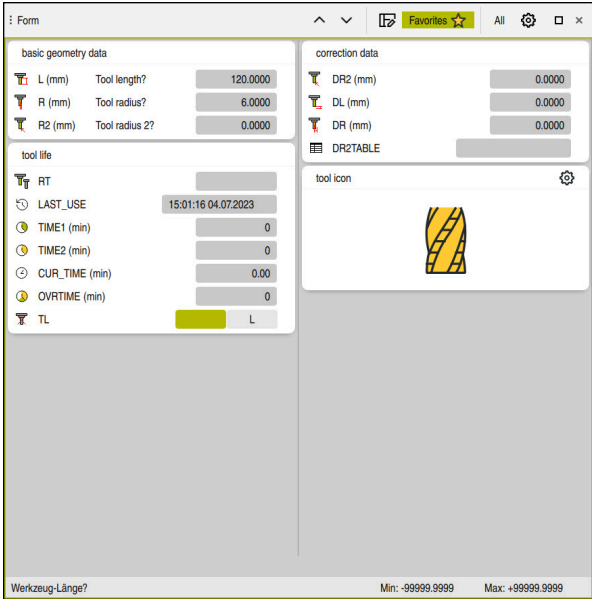
The **Form** workspace in the **Favorites** view

The control displays the following information for each parameter:

■ Icon of the parameter, if applicable
■ Parameter name
■ Unit of measure as needed
■ Parameter description
■ Current value

The control displays the contents of specific tables in groups within the **Form** workspace.

Refer to your machine manual.
If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Buttons and icons

The **Form** workspace contains the following buttons, icons or shortcuts:

Buttons, icons or shortcuts		Meaning
∧ SHIFT + UP	∨ SHIFT + DOWN	**Navigate** Navigate between table rows
🖉		**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
Favorites		In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites.
All		In this view the control shows all areas.
⚙		**Settings** ■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 1678 ■ Change the size of the graphic in the **Tool Icon** area
✚		**Add** The control only shows this icon when you are adapting the layout. With this icon you can add the following elements: ■ Column You can divide the workspace into several columns. **Further information:** "Adding a column in the workspace", Page 1678 ■ Area In the **Favorites** view you can add another area.
▬		**Remove** The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the parameter descriptions. The selected setting is modally effective.
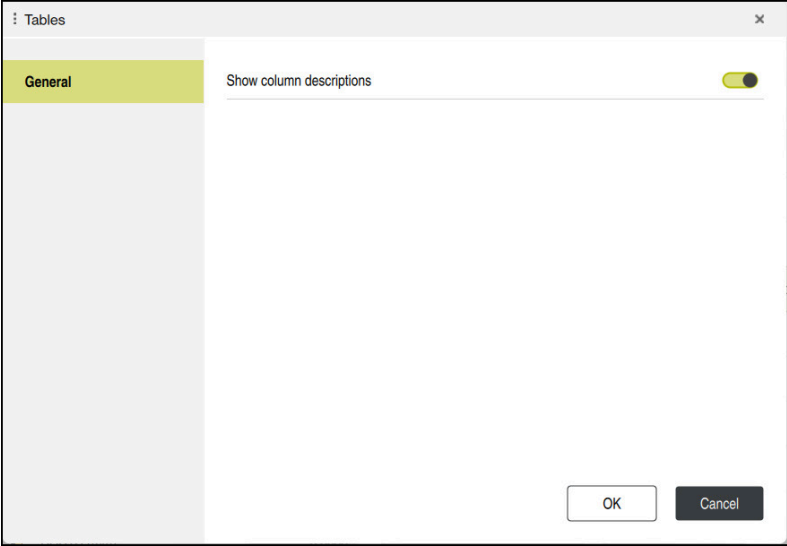
38.4.1 Adding a column in the workspace

To add a column:

	▶ Select **Configure the layout**
	> The control enables all functions for adapting the layout of the workspace.
	▶ In the workspace, swipe to the left
	▶ Select **Add**
	> The control adds a new column.
	▶ Move the areas if required
	▶ Select **Configure the layout**
	> The control saves your changes.

Notes

■ The control displays an icon of the selected tool type in the **Tool Icon** area.
Further information: "Tool types", Page 261

38.5 Accessing table values

38.5.1 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)
- Preset table ***.pr** (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 1680
TABDATA WRITE	Write a value to a table cell	Page 1681
TABDATA ADD	Add a value to a table value	Page 1683

If the unit of measure used in the NC program differs from that used in the table, the control converts the values from **millimeters** to **inches**, and vice versa.

Related topics

- Fundamentals regarding variables

  Further information: "Basics", Page 1006
- Tool table

  Further information: "Tool table tool.t", Page 1684
- Compensation tables

  Further information: "Compensation tables", Page 1732
- Reading values from freely definable tables

  Further information: "Reading a freely definable table with FN 28: TABREAD", Page 1041
- Writing values to freely definably tables

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1039

38.5.2 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 automatically converts 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**, **CORR-WPL** or **PRESET**	Read the value from the tool table or a compensation table ***.tco** or ***.wco** or from the preset table
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name

38.5.3 Writing table values with TABDATA WRITE

Application

Use the function **TABDATA WRITE** to write a value 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. Alternatively, you can define the value directly in the NC function **TABDATA WRITE**.

Input

11 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1	; Write the value from **Q1** to row 3, column **DR**, of the compensation table

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ Functions ▶ TABDATA ▶ TABDATA WRITE

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, **CORR-WPL** or **PRESET**	Write a value to a compensation table ***.tco** or ***.wco** or to the preset table
COLUMN	Column name
	Fixed or variable name
KEY	Row number
	Fixed or variable name
= or **SET UNDEFINED**	Write the table value or assign the status **undefined**
Number, **Name** or **QS**	Table value
	Fixed or variable number or name
	Only if **=** has been selected

Note

<div style="border:1px solid #000;">

<div style="background:#1a9ae0; color:#fff; text-align:center;">***NOTICE***</div>

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. If the previous value is kept, there is a danger of collision!

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

</div>

38.5.4 Adding table values with TABDATA ADD

Application

Use the **TABDATA ADD** function to add a value to an existing table value.

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. Alternatively, you can define the value directly in the NC function **TABDATA ADD**.

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 803

Input

11 TABDATA ADD CORR-TCS COLUMN "DR" KEY "3" = Q1	; Add the value from **Q1** to row 3, column **DR**, of the compensation table

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ Functions ▶ TABDATA ▶ TABDATA ADD

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, **CORR-WPL** or **PRESET**	Write a value to a compensation table ***.tco** or ***.wco** or to the preset table
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name
Number	Value to be added Fixed or variable number

38.6 Tool tables

38.6.1 Overview

This chapter describes the tool tables of the control.

- Tool table **tool.t**

  Further information: "Tool table tool.t", Page 1684
- Touch probe table **tchprobe.tp** (#17 / #1-05-1)

  Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 1694

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 269

38.6.2 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 269
- Tool data required for milling or drilling tools
  Further information: "Tool data for milling and drilling tools", Page 265

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 you to identify each tool unambiguously (e.g., for calling a tool).
	Further information: "Tool call by TOOL CALL", Page 279
	You can define an index after the period.
	Further information: "Indexed tool", Page 258
	This parameter applies to all tools, regardless of technology.
	Input: **0.0...32767.9**
NAME	**Tool name?**
	The tool name identifies a tool, for example when calling it.
	Further information: "Tool call by TOOL CALL", Page 279
	You can define an index after the period.
	Further information: "Indexed tool", Page 258
	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 reference point
	Further information: "Tool carrier reference point", Page 253
	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 253
	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, for example, ball-nose cutters or toroid cutters.
	Further information: "3D tool compensation (#9 / #4-01-1)", Page 805
	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 compensation values automatically after measuring the workpiece.
	Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
	Is added to the parameter **L**
	Input: **−999.9999...+999.9999**
DR	**Tool radius oversize?**

Parameter	Meaning
	Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291 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:** "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291 Is added to parameter **R2** Input: **-999.9999...+999.9999**
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 999 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 258 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.

Parameter	Meaning
	This parameter applies to all tools, regardless of technology.
	Input: **0...99999**
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 999
	The behavior depends on the machine. Refer to your machine manual.
	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 tool is cutting a workpiece when the spindle is switched on and the control moves the tool at the machining feed rate. The control counts this time automatically and enters the current tool age in minutes.
	You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring.
	The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.
	This parameter applies to all tools, regardless of technology.
	Input: **0...99999.99**
TYP	**Tool type?**
	Depending on the selected tool type, the control displays the suitable tool parameters in the **Form** workspace of the tool management.
	Further information: "Tool types", Page 261
	Further information: "Tool management ", Page 269
	This parameter applies to all tools, regardless of technology.
	Selection by means of a selection window
	Input: **MILL**, **MILL_R**, **MILL_F**, **MILL_FACE**, **BALL**, **TORUS**, **MILL_CHAMFER**, **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**

Parameter	Meaning
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**
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 (e.g., of 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, for example, necks of end mills or side milling cutters.
	The tool can contain a neck radius **RN** only if the useful length **LU** is longer than the **LCUTS** length of the cutting edge.
	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 for Tools (#17 / #1-05-1)", Page 1555
	Further information: "Cutting data calculator", Page 1181
	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 1724
	Selection by means of a selection window
	Input: **Text width 32**
CUTDATA	**Cutting data table?**
	Further information: "Cutting data calculator", Page 1181
	Select the cutting data table with the ***.cut** or ***.cutd** file extension for cutting data calculation.
	Further information: "Cutting data table *.cut", Page 1725
	Selection by means of a selection window
	Entry: **Text width 20**
LTOL	**Wear tolerance: length?**
	Permitted tool length deviation in wear detection for automatic tool measurement.
	Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
	If the entered value is exceeded, the control locks the tool in column **TL**.
	Input: **0.0000...5.0000**

Parameter	Meaning
RTOL	**Wear tolerance: radius?**
	Permitted tool radius deviation in wear detection for automatic tool measurement.
	Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
	If the entered value is exceeded, the control locks the tool in column **TL**.
	Input: **0.0000...5.0000**
R2TOL	**Wear tolerance: Radius 2?**
	Permitted tool radius 2 deviation in wear detection for automatic tool measurement.
	Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
	If the entered value is exceeded, the control locks the tool in column **TL**.
	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 for Tools (#17 / #1-05-1)", Page 1555
	Input: **-**, **+**
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 for Tools (#17 / #1-05-1)", Page 1555
	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 for Tools (#17 / #1-05-1)", Page 1555
	Is added to the machine parameter **offsetToolAxis** (no. 122707)
	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 for Tools (#17 / #1-05-1)", Page 1555
	If the entered value is exceeded, the control locks the tool in column **TL**.
	Input: **0.0000...9.0000**
RBREAK	**Breakage tolerance: radius?**
	Permitted tool radius deviation in breakage detection for automatic tool measurement.

Parameter	Meaning
	Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
	If the entered value is exceeded, the control locks the tool in column **TL**.
	Input: **0.0000...9.0000**
NMAX	**Maximum speed [rpm]**
	Limitation of spindle speed for the programmed value including control by the potentiometer.
	Input: **0...999999**
LIFTOFF	**Lift-off allowed?**
	Allow automatic tool lift-off with active **M148** or **FUNCTION LIFTOFF**:
	■ **Y**: Activate **LIFTOFF**
	■ **N**: Deactivate **LIFTOFF**
	Further information: "Automatically lifting off upon an NC stop or a power failure with M148", Page 996
	Further information: "Automatic tool liftoff with FUNCTION LIFTOFF", Page 881
	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 (#17 / #1-05-1)", Page 1694
	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 drills, for example.
	Further information: "Cycles for Drilling, Centering and Thread Machining", Page 445
	Input: **-180...+180**
LAST_USE	**Date/time of last tool usage**
	The time at which the tool was last used
	The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.
	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 1698
	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 (#45 / #2-31-1)) from the **AFC.tab** table
	Further information: "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

Parameter	Meaning
	Selection by means of a selection window
	Entry: **Text width 10**
ACC	**ACC active?**
	Activate or deactivate active chatter control (ACC (#145 / #2-30-1)):
	■ **Y**: Activate
	■ **N**: Deactivate
	Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 893
	Selection by means of a selection window
	Input: **Y**, **N**
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 Drilling, Centering and Thread Machining", Page 445
	Input: **-9.9999...+9.9999**
AFC-LOAD	**Reference power for AFC [%]**
	Tool-dependent reference power for AFC (#45 / #2-31-1).
	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 890
	Input: **1.0...100.0**
AFC-OVLD1	**AFC overload warning level [%]**
	Cut-related tool wear monitoring for AFC (#45 / #2-31-1).
	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 892
	Input: **0.0...100.0**
AFC-OVL2	**AFC ovrload switch-off level [%]**
	Cut-related tool load monitoring for AFC (#45 / #2-31-1).
	The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect.
	If this column contains a value, the control will ignore the column **AFC-OVLD1**.
	Further information: "Monitoring tool wear and tool load", Page 892
	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 273
	Selection by means of a selection window
	This parameter applies to all tools, regardless of technology.
	Entry: **Text width 20**
TSHAPE	**3D tool model**
	Assigning a 3D model for exact definition of the tool for graphical simulation and collision monitoring.
	Further information: "Tool model (#140 / #5-03-2)", Page 277

Parameter	Meaning
	Selection by means of a selection window
	Input: **Text width 50**
DR2TABLE	**Compensation val. table for DR2**
	Assigning a compensation value table ***.3dtc** for 3D tool radius compensation depending on the contact angle. This allows the control to compensate for inaccuracies in the shape of a ball-nose cutter or the deflection behavior of a touch probe, for example.
	Selection by means of a selection window
	Entry: **Text width 16**
OVRTIME	**Tool life expired**
	Time in minutes during which the tool may be used beyond the tool life defined in column **TIME2**.
	The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control will use 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**
DB_ID	**ID for central tool management**
	The database-ID allows you to identify a tool (e.g., within a tool management system by using client applications).
	Further information: "Database ID", Page 257
	For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.
	Further information: "Indexed tool", Page 258
	This parameter applies to all tools, regardless of technology.
	Input: **Text width 40**
R_TIP	**Radius at the tip**
	Radius at the tool tip for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of tools such as countersinks.
	Input: **0.0000...999.9999**

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 1698

- 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 changes only the position of the tool in the simulation.

- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look 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 the 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.

38.6.3 Touch probe table tchprobe.tp (#17 / #1-05-1)

Application

The touch probe table **tchprobe.tp** defines the touch probe and data for the probing process, such as 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 269
- Touch probe functions

  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257
- Calibrating touch probe cycles for the workpiece touch probe

  Further information: "Calibrating a workpiece touch probe (#17 / #1-05-1)", Page 1231
- Calibrating touch probe cycles for the tool touch probe

  Further information: "Calibrating a tool touch probe (#17 / #1-05-1)", Page 1249
- Automatic touch probe cycles for the workpiece

  Further information: "Touch-Probe Cycles for Workpieces (#17 / #1-05-1)", Page 1291
- Automatic touch probe cycles for the tool

  Further information: "Touch-Probe Cycles for Tools (#17 / #1-05-1)", Page 1555
- Automatic touch probe cycles for measuring the kinematics

  Further information: "Touch-Probe Cycles for Kinematics Measuring", Page 1575

Description of function

NOTICE
Danger of collision!
The control cannot protect L-shaped styli from collisions using Dynamic Collision Monitoring DCM (#40 / #5-03-1). When using a touch probe with an L-shaped stylus there is a risk of collision!
▶ Carefully run in the NC program or program section in the **Program Run Single Block** operating mode
▶ Watch out for possible collisions!

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 following values are available for the TS 642 touch probe:
	▪ **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]**
	According to the selection of the **STYLUS** column, this parameter has the following function:
	▪ **SIMPLE**: Offset of the touch probe axis to the spindle axis in the main axis
	▪ **L-TYPE**: Length of extension on an L-shaped stylus
	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?**
	According to the selection of the **STYLUS** column, this parameter has the following function:
	▪ **SIMPLE**: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible).
	▪ **L-TYPE**: The control orients the extension using the spindle angle.
	Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible).
	Input: **0.0000...359.9999**

Parameter	Meaning
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: **+10...+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**
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**
STYLUS	**Shape of the stylus**

Parameter	Meaning
	■ **SIMPLE**: Straight stylus
	■ **L-TYPE**: L-shaped stylus

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
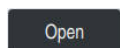
▶ Select **Open**
> The control opens the **Touch probes** application.
▶ Activate **Edit**
▶ Select the desired value
▶ Edit the value

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.

38.6.4 Creating a tool table in inches

To create a tool table in inches:

▶ Select the **Manual** operating mode

▶ 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 **Create new table**
> The control opens the **Create new table** window.
▶ Select a folder with the corresponding table type (e.g., **t**)
▶ Select the desired prototype

▶ Select a path
> The control opens the **Save as** window.
▶ Select the **table** folder
▶ Enter a name (e.g., **tool**)
▶ Select **Create** twice
> The control opens the **Tool table** tab in the **Tables** operating mode.
▶ Restart the control

▶ Acknowledge **Power interrupted** with the **CE** key

▶ Select the **Tool table** tab in the **Tables** operating mode
> The control uses the newly created table as a tool table.

> ℹ️ To use the **Tool management** application you have to create all existing tool tables in inches.

38.7 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 279
■ Tool table
   Further information: "Tool table tool.t", Page 1684

Requirement

■ The tool is defined in the tool management.
   Further information: "Tool management ", Page 269

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
	With the machine parameter **deleteLoadedTool** (no. 125301) you define whether you are allowed to edit the **T** column. The machine manufacturer enables this parameter.
	Further information: "Tool table tool.t", Page 1684
	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 1684
	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**
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?**

Parameter	Meaning
	The control automatically loads the tool comment from the tool table. **Further information:** "Tool table tool.t", Page 1684 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**
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 1684 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**

Parameter	Meaning
	Value for evaluation in the PLC
	The machine manufacturer defines the function of this parameter. Refer to your machine manual.
	Entry: **Text width 16**

38.8 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 287
- Working with a pallet table
  Further information: "Pallet Machining and Job Lists", Page 1619
- Tool data from the tool table
  Further information: "Tool table tool.t", Page 1684

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
  In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.
  Further information: "Creating the tool usage file", Page 287
- The **Generate tool-usage file** setting is set to **Once** or **Always**
  Further information: "Channel Settings", Page 1782

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**
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**.

Parameter	Meaning
	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 saves the tool usage file as a dependent file (***.dep**).

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: "Areas of file management", Page 825

38.9 T usage order (#93 / #2-03-1)

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 Advanced Tool Management (#93 / #2-03-1)
- Tool-usage file has been created
  Further information: "Creating the tool usage file", Page 287
  Further information: "Tool usage file", Page 1702

Description of function

When you select an NC program in the **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 258
	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 258
	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 1698
	■ **T no. missing**: Tool is not defined in the tool management
	Further information: "Tool management ", Page 269
T PROG	Number or name of the programmed tool, including an index as needed
	Further information: "Indexed tool", Page 258
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 1702
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 1702
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 1698

38.10 Tooling list (#93 / #2-03-1)

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 Advanced Tool Management (#93 / #2-03-1)
- Tool-usage file has been created
  Further information: "Creating the tool usage file", Page 287
  Further information: "Tool usage file", Page 1702

Description of function

When you select an NC program in the **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 numerical 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 258 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 1698 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 273
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 258
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 1702
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 1698

38.11 Freely definable tables *.tab

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: "NC functions for freely definable tables", Page 1039

Description of function
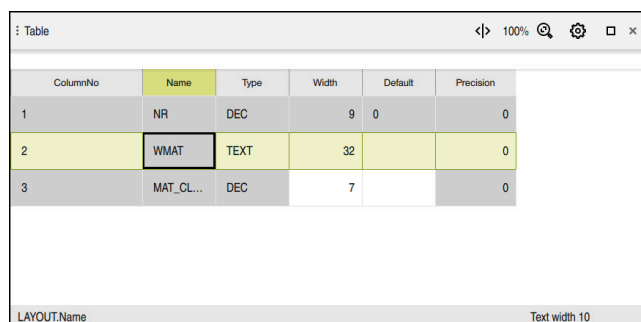
When you create a freely definable table, the control will provide various table templates for selection.

The machine manufacturers can create their own table templates and store them in the control.

After you have created a freely definable table, you can modify its properties. you modify the table properties in the **LAYOUT** application.

Further information: "Modifying the properties of freely definable tables", Page 1709

In the **LAYOUT** application, the control shows the columns of the table row by row.
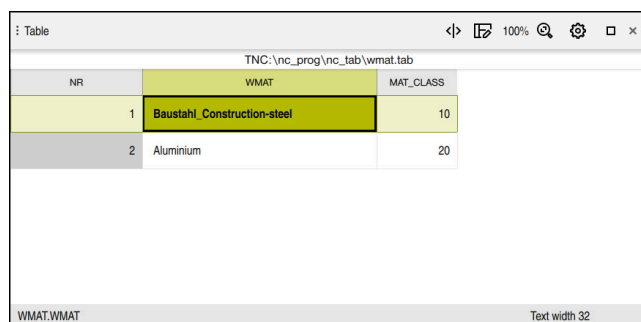
Freely definable table in the **LAYOUT** application

Freely definable table in the **Table** workspace

Properties of a table column

When you change any table properties, each column has the following properties:

Column	Meaning
Name	Name of the column
Width	Maximum number of characters in the column
Default	Default value of each new row Optional input

Type	The control offers the following possible selections in the **Type** column:
	
	■ **TEXT**: Text entry
	■ **SIGN**: Algebraic sign + or –
	■ **BIN**: Binary number
	■ **DEC**: Positive integer
	■ **HEX**: Hexadecimal number
	■ **INT**: Integer
	■ **LENGTH**: Floating-point number (mm or inch)
	
	   ❗ If you write values from an inch program to a freely definable table, the control converts the values.
	
	   ❗ If the unit of measure is inches, then the column has one more decimal place than you define.
	
	■ **FEED**: Feed rate (mm/min or 0.1 ipm)
	■ **IFEED**: Feed rate (mm/min or ipm)
	
	   ❗ If the unit of measure is inches, then the column has one more decimal place than you define.
	
	■ **FLOAT**: Floating-point number
	■ **BOOL**: Logical value
	■ **INDEX**: Index
	■ **TSTAMP**: Time and date with the format HH:MM:SS DD.MM.YYYY
	■ **UPTEXT**: Text entry in capital letters
	■ **PATHNAME**: Path name
	
	   ❗ In the columns with the data types **BIN**, **DEC** or **HEX** you can enter the values as binary numbers, positive integers or hexadecimal numbers. The control converts the entered values into the column's respective data type.

Precision	Maximum number of decimal places

38.11.1 Modifying the properties of freely definable tables

To insert a new column:

▶ Open an empty freely definable table

▶ Select **Edit table characteristics**

> The control opens the **LAYOUT** application.

▶ Activate **Editing**

▶ Select **Insert rows**

> The control opens the **Insert rows** window.

▶ Enter **Column name**

▶ Select **Column type**

> The control displays a selection menu.

> **i** You cannot change the column name or column type later.

▶ Select the desired column type

Further information: "Properties of a table column", Page 1708

▶ Select **OK**

> The control inserts a new row at the end of the table.

▶ In the **Width** column you define the maximum number of characters per column (e.g., **12**).

▶ Define a value in the **Default** if needed.

▶ In the **Precision** column you define the number of decimal places (e.g., **3**).

▶ Select **Save changes**

> The control opens the **Save layout changes** window.

▶ Select **OK**

> The control closes the **LAYOUT** application.
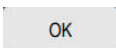
Notes

■ 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 1066

■ The sequence of columns in the **Table** workspace is independent of the sequence of rows in the **LAYOUT** application. You can edit the sequence of columns in the **Table** workspace.

Further information: "Settings in the Table workspace", Page 1674

38.12 Preset table *.pr

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 196

Related topics

■ Setting and activating presets

Further information: "Preset management", Page 705

Description of function

By default, the preset table has the name **preset.pr**, and is saved in the **TNC:\table** directory. The preset table is open in the **Tables** operating mode by default.

> ⚙ Refer to your machine manual.
>
> The machine manufacturer 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.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
⊕	Active row
🔒	Write-protected row

When you define a preset, the control opens a window with the following input options:

Icon or button	Function
⊹	**actual position capture** The control opens or closes the position display of the status overview. When you select an axis, the control applies the selected value at **Set a preset**. **Further information:** "actual position capture in the preset table", Page 1715
Set a preset	The control interprets the entered value as desired display value for the actual position. The control calculates the required table value from this. The entered value is active in the basic coordinate system **B-CS**. **Further information:** "Basic coordinate system B-CS", Page 695 When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value. The entered value is active incrementally in the basic coordinate system **B-CS**.
Edit	The control accepts the entered value unchanged as table value. The entered value refers to the coordinate origin of the basic coordinate system **B-CS**.

Parameters of the preset table

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 695 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 695 Input: **-99999.99999...+99999.99999**
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 695 Input: **-99999.99999...+99999.99999**
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 695 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 707 Input: **-99999.9999999...+99999.9999999**
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 695 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 707 Input: **-99999.9999999...+99999.9999999**
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 695 Has the effect of a basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 707 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 692 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 692 Input: **-99999.99999...+99999.99999**
Z_OFFS	Position of the Z axis for the preset

Parameter	Meaning
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	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 692
	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 692
	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 692
	Input: **-99999.9999999…+99999.9999999**
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 692
	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 692
	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 692
	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.

Basic transformation and offset

The control interprets the basic transformations **SPA**, **SPB** and **SPC** as basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. During program execution, the control moves the linear axes in accordance with the basic rotation without any change in the workpiece position.

Further information: "Basic rotation and 3D basic rotation", Page 707

The control interprets all offsets for each respective axis as a shift in the machine coordinate system **M-CS**. The effect that offsets have is contingent on the kinematics.

Further information: "Machine coordinate system M-CS", Page 692

> **i** HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** probing function to determine the misalignment of a workpiece. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating the rotation of a workpiece", Page 1270

Calculated results	Actual value	Nominal value
🟢 Basic rotation	-360	0
⚫ Table rotation	0	0.00000
Compensate the active preset	Align rotary table	Correct the pallet reference point

Results of the **Rotation (ROT)** probing function

If you activate the **Basic rotation** toggle switch, the control interprets the misalignment as a basic transformation. When using the **Compensate the active preset** button, the control saves the result in the columns **SPA**, **SPB** and **SPC** of the preset table. The **Align rotary table** button has no function in this case.

If you activate the **Table rotation** toggle switch, the control interprets the misalignment as an offset. When using the **Compensate the active preset** button, the control saves the result in the columns **A_OFFS**, **B_OFFS** and **C_OFFS** of the preset table. To move the rotary axes to the position of the offset, use the **Align rotary table** button.

Write-protection for table rows

The **Lock record** button allows protecting any rows of the preset table against overwriting. The control enters the value **L** in the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 1716

Alternatively, the row can be protected with a password. The control enters the value **###** into the **LOCKED** column.

Further information: "Protecting table rows with a password", Page 1716

The control displays an icon ahead of write-protected rows.

ⓘ If the control displays the value **OEM** in the **LOCKED** column, this column has been locked by the machine manufacturer.

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 would lock the protected rows permanently.

▶ Protecting table rows without a password is recommended

▶ Note down your passwords

38.12.1 actual position capture in the preset table

To load the actual position of an axis into the preset table:

| Edit | ▶ Activate the **Edit** toggle switch |

▶ Double-tap or double-click the table row to be changed (e.g., in the **X** column)

> The control opens a window with input options.

▶ Select **actual position capture**

> The control opens the position display of the status overview.

▶ Select the desired value

> The control loads the value into the window and activates the **Set a preset** button.
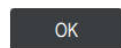
| OK | ▶ Select **OK** |

> The control calculates the table value that is needed and enters the value in the table.

▶ If required, close the position display of the status overview

38.12.2 Activating write protection

Protecting table rows without a password

To protect a table row without a password:

Edit	▶ Activate the **Edit** toggle switch
	▶ Select the desired row
Lock record	▶ Activate the **Lock record** toggle switch
	> The control enters the value **L** in the **LOCKED** column.
🔓	> The control activates write-protection and displays an icon ahead 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 would lock the protected rows permanently.
>
> ▶ Protecting table rows without a password is recommended
> ▶ Note down your passwords

To protect a table row with a password:

Edit	▶ Activate the **Edit** toggle switch
	▶ Double-tap or double-click the **LOCKED** column of the desired row
	▶ Enter the password
	▶ Confirm your input
	> The control enters the value **###** in the **LOCKED** column.
🔓	> The control activates write-protection and displays an icon ahead of the row.

38.12.3 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

Edit	▶ Activate the **Edit** toggle switch
Lock record	▶ Deactivate the **Lock record** toggle switch
	> The control removes the value **L** from the **LOCKED** column.
	> The control deactivates the write protection and removes the icon ahead 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 would lock 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:

Edit ▶ Activate the **Edit** toggle switch

▶ Double-tap or double-click the **LOCKED** column of the desired row
▶ Delete **###**
▶ Enter the password
▶ Confirm your input
▷ The control deactivates write-protection and removes the icon ahead of the row.

38.12.4 Creating a preset table in inches

If you define inches as the unit of measure in the **Machine Settings** menu item, the unit of measure of the preset table will not be adjusted automatically.

Further information: "The Machine Settings menu item", Page 1781

To create a preset table in inches:

Shut down	▶ Restart the control
	▶ Do not acknowledge the **Power interrupted** message
📁	▶ Select the **Files** operating mode
	▶ Open the **TNC:\table** folder
	▶ Rename the original file (e.g., **preset.pr** as **preset_mm.pr**)
⊞	▶ Select the **Tables** operating mode
⊞	▶ Select **Create new table**
	> The control opens the **Create new table** window.
	▶ Select the **pr** folder
📄	▶ Select the desired prototype
	▶ Select a path
	> The control opens the **Save as** window.
	▶ Select the **table** folder
	▶ Enter the name **preset.pr**
Create	▶ Select **Create** twice
	> The control opens the **Presets tab in the Tables** operating mode.
Shut down	▶ Restart the control
CE	▶ Acknowledge **Power interrupted** with the **CE** key
⊞	▶ Select the **Presets** tab in the **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. If the previous value is kept, there is a danger of collision!

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

- 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.
- If you edit the value of the **DOC** column, then the preset must be reactivated. Only then does the control apply the new value.
  Further information: "Activating presets", Page 707
- 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 1635
- If a manual probing process or an NC program is interrupted or stopped, you cannot edit the preset table. When you double-tap or double-click a table cell the control shows the **Editing not possible. Perform internal stop?** window. If you select **Yes**, the control may lose touch points or modally active program information.

Notes about machine parameters

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- If the unit of measure of the preset table does not match the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the dialog bar of the **Tables** operating mode.
- The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offsets in the following NC functions:
  - **FUNCTION PARAXCOMP**
    Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 933
  - **FUNCTION POLARKIN** (#8 / #1-01-1)
    Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 940
  - **FUNCTION TCPM** or **M128** (#9 / #4-01-1)
    Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 787

38.13 Point table *.pnt

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 391

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**

38.13.1 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

Edit
    ▶ 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 block** toggle switch to skip this point in **Program Run** operating mode.
Further information: "Icons and buttons", Page 1640

38.14 Datum table *.d

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics
- Contents and creation of a datum table
  Further information: "Datum table *.d", Page 1721
- Editing a datum table during a program run
  Further information: "Compensation during program run", Page 1658
- Preset table
  Further information: "Preset table *.pr", Page 1710

Description of function

The values of the columns **X**, **Y** and **Z** are applied as shifts in the workpiece coordinate system **W-CS**. The values of the columns **A**, **B**, **C**, **U**, **V** and **W** are applied as shifts in the machine coordinate system **M-CS**.
Further information: "Comparison of offset and 3D basic rotation", Page 1288

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
	Transformation relating to the workpiece coordinate system **W-CS**
	Further information: "Workpiece coordinate system W-CS", Page 697
	Input: **-99999.99999...+99999.99999**
Y	Y coordinate of the datum
	Transformation relating to the workpiece coordinate system **W-CS**
	Further information: "Workpiece coordinate system W-CS", Page 697
	Input: **-99999.99999...+99999.99999**
Z	Z coordinate of the datum
	Transformation relating to the workpiece coordinate system **W-CS**
	Further information: "Workpiece coordinate system W-CS", Page 697
	Input: **-99999.99999...+99999.99999**
A	Axis angle of the A axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-360.0000000...+360.0000000**
B	Axis angle of the B axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-360.0000000...+360.0000000**
C	Axis angle of the C axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-360.0000000...+360.0000000**
U	Position of the U axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-99999.99999...+99999.99999**
V	Position of the V axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-99999.99999...+99999.99999**
W	Position of the W axis for the datum
	Offset relating to the machine coordinate system **M-CS**
	Further information: "Machine coordinate system M-CS", Page 692
	Input: **-99999.99999...+99999.99999**
DOC	**Comment on shift?**
	Input: **Text width 15**

38.14.1 Editing a datum table

You can edit the active datum table during program run.

Further information: "Compensation during program run", Page 1658

To edit a datum table:

Edit	▶ Activate **Edit**
	▶ Select the value
	▶ Edit the value
	▶ Save the edited value, for example 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

38.15 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 1724
- Table for tool materials **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 1724
- Cutting data table ***.cut**

  Further information: "Cutting data table *.cut", Page 1725
- Diameter-dependent cutting data table ***.cutd**

  Further information: "Diameter-dependent cutting data table *.cutd", Page 1726

Related topics

- Cutting data calculator

  Further information: "Cutting data calculator", Page 1181
- Tool management

  Further information: "Tool management ", Page 269

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 **WMAT.tab** table **Further information:** "Table for workpiece materials WMAT.tab", Page 1724 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 1724 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 283 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 284 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 **WMAT.tab** table
	Further information: "Table for workpiece materials WMAT.tab", Page 1724
	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 1724
	Selection by means of a selection window
	Input: **Text width 32**
VC	Cutting speed in m/min
	Further information: "Cutting data", Page 283
	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 284
	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.

38.16 Pallet table *.p

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: "The Job list workspace", Page 1620
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1630

Description of function

Pallet tables can be opened in the **Tables**, **Editor**, and **Program Run** operating modes. In the **Editor** and **Program Run** operating modes, the control opens the pallet table in the **Job list** workspace and not 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 1649
	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**
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, the entry is mandatory.
	Corresponds to the **Machinable** toggle switch in the **Form** workspace.
	Selection using a selection menu
	Input: No value, **MA**
LOCK	Locked?

Parameter	Meaning
	Using an ***** you can exclude the row of the pallet table from execution. Press the **ENT** key to identify the row with the entry *****. 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 executed.
	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 1630
	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**
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 1630
	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 1630
	Input: **Text width 8**
SP-X	**Clearance height?**
	Clearance height in the X axis for tool-oriented machining
	Further information: "Tool-oriented machining", Page 1630

Parameter	Meaning
	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 1630
	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 1630
	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 1630
	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 1630
	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 1630
	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 1630
	Input: **–999999.99999…+999999.99999**
SP-V	**Clearance height?**
	Clearance height in the V axis for tool-oriented machining
	Further information: "Tool-oriented machining", Page 1630
	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 1630
	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 1620
	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 1620

Parameter	Meaning
	Input: **0...99999**

38.17 Compensation tables

38.17.1 Overview

The control provides the following compensation tables:

Table	Further information
Compensation table ***.tco** Compensation in the tool coordinate system **T-CS**	Page 1732
Compensation table ***.wco** Compensation in the working plane coordinate system **WPL-CS**	Page 1734

38.17.2 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 801
- Contents of the compensation table ***.wco**
  Further information: "Compensation table *.wco", Page 1734
- Editing compensation tables during program run
  Further information: "Compensation during program run", Page 1658
- Tool coordinate system **T-CS**
  Further information: "Tool coordinate system T-CS", Page 703

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 active tool, the control will display 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**

38.17.3 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**.

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 801
- Contents of the compensation table ***.tco**
  Further information: "Compensation table *.tco", Page 1732
- Editing compensation tables during program run
  Further information: "Compensation during program run", Page 1658
- Working plane coordinate system **WPL-CS**
  Further information: "Working plane coordinate system WPL-CS", Page 699

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**

38.18 Tables for AFC (#45 / #2-31-1)

38.18.1 Basic AFC settings in **AFC.tab**

Application

In the **AFC.tab** table, you define 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) (#45 / #2-31-1)", Page 884

Requirement

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))

Description of function

The data in this table are default values that, during a teach-in cut, are copied into an associated dependent file of the relevant NC program. The values are the basis for feedback control.

Further information: "Description of function", Page 1738

> **i** If you define a tool-specific reference power in the **AFC-LOAD** column in the tool table, the control will create the associated dependent file for the respective 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 response
	Enter the value in percent of the programmed feed rate
	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 892
	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
	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 892
	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
	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
	Input: **0…999**
OVLD	Desired reaction of the control to overload:
	■ **M**: Execution of a macro defined by the machine manufacturer
	■ **S**: Immediate NC stop
	■ **F**: Execute NC stop when the tool is no longer in the material
	■ **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 while feedback control is active, the control will conduct an overload reaction.
	In conjunction with the cut-related tool wear monitoring function, the control will evaluate only the options **M**, **E**, and **L**!
	For tool-load monitoring with the column **AFC_OVLD2**, this parameter has no function.
	Input: **M**, **S**, **F**, **E**, **L**, or **-**
POUT	Spindle power at which the control will detect that the tool exits the workpiece
	Enter the value in percent of the learned reference load

Parameter	Meaning
	Recommended input value: 8%
	Input: **0…100**
SENS	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.
	Input: **0…999**
PLC	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**

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 1066

38.18.2 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**. The string **<name>** is identical to 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 1734
- Setting up and using AFC

  Further information: "Adaptive feed control (AFC) (#45 / #2-31-1)", Page 884

Requirement

- Adaptive Feed Control software option (AFC (#45 / #2-31-1))

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 1734

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)
	i In conjunction with AFC (#45 / #2-31-1), the following characters are not permitted in the tool name: # $ & , .
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 is recorded for this machining step. The control overwrites any existing values in this row ■ **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 settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: "Areas of file management", Page 825

38.18.3 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. The string **<name>** is identical to 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) (#45 / #2-31-1)", Page 884

Requirement

■ Adaptive Feed Control software option (AFC (#45 / #2-31-1))

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 manufacturer has been run ■ **S:** Immediate NC stop was conducted ■ **F**: NC stop was performed once the tool was no longer in the material ■ **E**: An error message was displayed ■ **L**: The current tool was locked ■ **-**: There was no overload response
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 settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.
Further information: "Areas of file management", Page 825

38.18.4 Editing the tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of 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 displays all the tables available for this NC program.
▶ Select a file (e.g., **AFC.TAB**)
> The control opens the file in the **Tables** operating mode.

39

Electronic Handwheel

39.1 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 190
- Handwheel superimpositioning with **M118** (#21 / #4-02-1)
  Further information: "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979
- Touch probe functions in **Manual** (#17 / #1-05-1) operating mode
  Further information: "Touch Probe Functions in the Manual Operating Mode (#17 / #1-05-1)", Page 1257

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 550 FS: 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 550 FS 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** toggle 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 you select **Manual** operating mode, the control deactivates the handwheel.

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 you have pressed the **NC Stop** key: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After you have pressed the **NC Stop** key: Manual traverse of 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 1657

- Switch on/off the "Tilt working plane" function (handwheel soft keys **MOP** and then **3D**)

Operating elements of an electronic handwheel
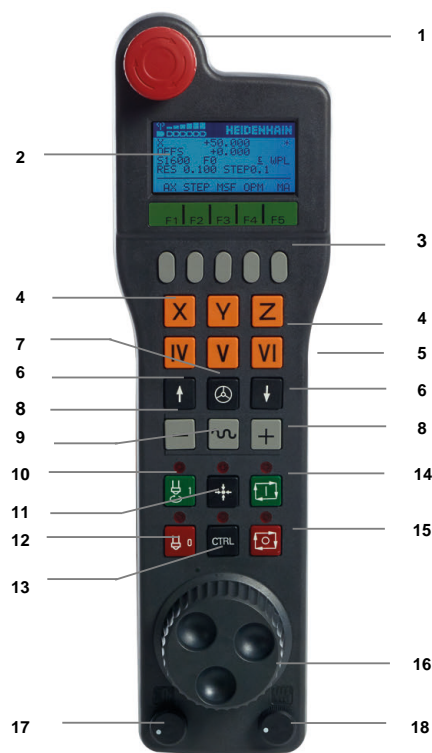
An electronic handwheel provides the following operating elements:

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

   You can activate or deactivate the handwheel.

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
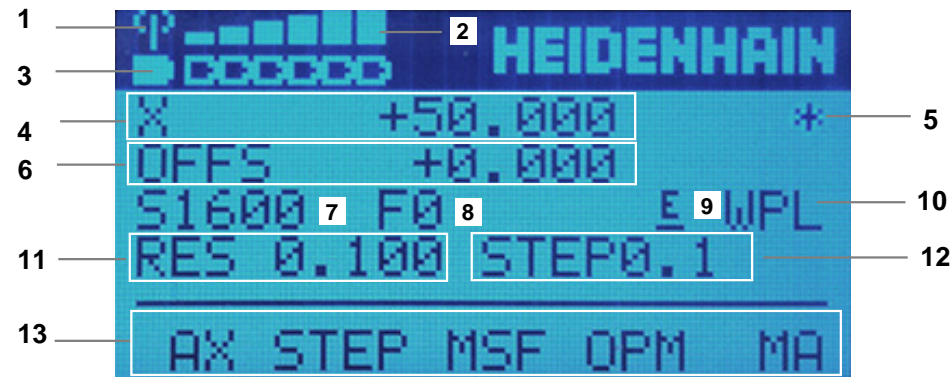
The display of an electronic handwheel consists of the following areas:

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   *****: Control in operation; program run has been started or axis is in motion

6   Handwheel superimpositioning from **M118** (#21 / #4-02-1)
    Further information: "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 979

7   **S1000**: Current spindle speed

8   Feed rate at which the selected axis is moving
    The control displays the current contouring feed rate while the program is running.

9   **E**: Error message
    If an error message appears on the control, the handwheel display shows the **ERROR** message 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**: function **Tool axis**
    ■ **WP**: function **Basic rotation**
    ■ **WPL**: **3D ROT** function
    Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 781

11 Handwheel resolution
    Distance that the selected axis moves per handwheel revolution
    Further information: "Handwheel resolution", Page 1747

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 1749

- **STEP**: Incremental jog positioning

  Further information: "Incremental jog positioning", Page 1749

- **MSF:** Execute various functions of the **Manual** operating mode (e.g., entering the feed rate **F**)

  Further information: "Entering miscellaneous functions M", Page 1749

- **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 the **Program Run** operating mode

- **MA**: Switching the magazine pockets

Handwheel resolution

The handwheel sensitivity specifies the distance an axis moves per handwheel revolution. 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 inches/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 inches/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 automatically activates the feed-rate potentiometer of the handwheel. When you deactivate the handwheel, the control automatically activates the feed-rate potentiometer of the machine operating panel.

In order to make sure that the feed rate does not increase while you switch between the potentiometers, the feed rate is either frozen or reduced.

If the feed rate before switching is higher than the feed rate after switching, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching is less than the feed rate after switching, 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.

39.1.1 Entering spindle speed S

To enter the spindle speed **S** by using an electronic handwheel:

▶ Press the handwheel soft key **F3** (**MSF**)
▶ Press the handwheel soft key **F2** (**S**)
▶ Select the desired spindle 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, 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**.

39.1.2 Entering the feed rate F

To enter the feed rate **F** by using an electronic handwheel:

▶ Press the handwheel soft key **F3** (**MSF**)
▶ Press the handwheel soft key **F3** (**F**)
▶ Select the desired feed rate by pressing the **F1** or **F2** key
▶ Load the new feed rate F with the handwheel soft key **F3** (**OK**)

> ⓘ 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**.

39.1.3   Entering miscellaneous functions M

To enter a miscellaneous function by using an electronic handwheel:

▶ Press the handwheel soft key **F3** (**MSF**)
▶ Press the handwheel soft key **F1** (**M**)
▶ 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 965

39.1.4   Creating a positioning block

> ⚙ Refer to your machine manual.
> Your machine manufacturer 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.

39.1.5   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

> ℹ 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.

> ℹ 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**.

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.

■ If the handwheel is active, the control shows a symbol next to the selected axis in the **Positions** workspace. The symbol indicates whether you can move the axis with the handwheel.

   Further information: "The Positions workspace", Page 149

> ⚙ Refer to your machine manual.
>
>   The machine manufacturer defines which axes you can move with the handwheel.

39.2 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.

HRA 551FS handwheel holder

HR 550FS handwheel

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.

> **i** 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 1755

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)

39.3 The 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 1741

■ HR 550FS wireless handwheel

Further information: "HR 550FS wireless handwheel", Page 1751

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

The Configuration area

In the **Configuration** area, the control displays different types of information about the connected wireless handwheel, such as the serial number.

The Statistics area

In the **Statistics** area, the control displays information about the transmission quality.

If the received signal quality is impaired and no longer ensures a perfect, safe stop of the axes, the wireless handwheel will perform an emergency stop.

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 1755
Further information: "Selecting the transmission power", Page 1754

The 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.

39.3.1 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.

39.3.2 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.

39.3.3 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.
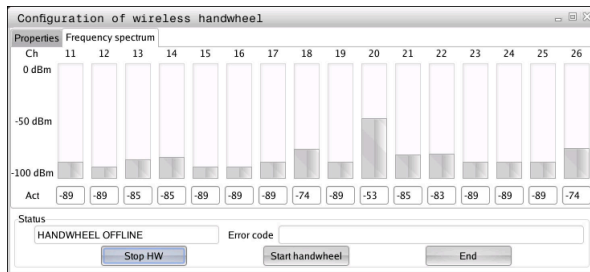
To set the radio channel manually:

⚒🖋 ▶ 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

> ℹ 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.

39.3.4 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

40 Override Controller

Application

The override controller is an operating element with additional functions compared to a usual override potentiometer.

In conjunction with the override controller, the control gives you the following possibilities:

- Use the dial to manipulate the feed rate and/or rapid traverse
- Start NC programs with the integrated **NC Start** button
- Receive tactile responses through vibrations
- Use breakpoints to define conditional stops
- Resume the NC program by increasing the override

Requirements

- Override controller OC 310

  The availability of the Override Controller depends on the machine.

  Refer to your machine manual.

- Control is fully booted

  The control only detects the override controller once the machine control voltage has been acknowledged.

- Tool inspection has been performed

  Further information: "The Tool check column in the Program workspace", Page 288

Description of function

Elements of the override controller
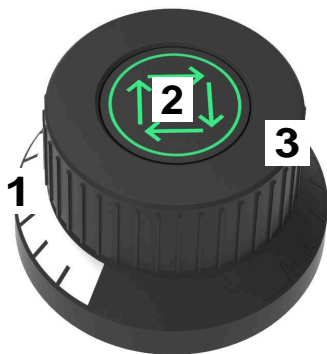
The override controller consists of the following elements:

1   Override scale

The override scale is illuminated in color up to the current override value.

Further information: "Visual feedback from the override controller", Page 1757

2   The **NC Start** button

The **NC Start** button starts the NC program.

Depending on the setting in the **Program run options** window, the NC program can be continued with the **NC Start** button.

3   Dial

Use the dial to change the override for the feed rate and/or rapid traverse.

Depending on the setting in the **Program run options** window, the NC program can be continued with the Override.

Visual feedback from the override controller

The override controller uses the following visual feedback:

Status	Override scale
Override Controller not active (e.g., because of an emergency stop)	Not illuminated
Override value of 0%	Not illuminated
Override value between 0% and 99.5%	White
Override value of 100%	Green
Override value greater than 100.5%	Blue

The **NC Start** button lights up green. The actual color may differ, depending on the machine.

Tactile feedback from the override controller

The override controller uses the following tactile feedback:

Status	Acknowledgment
Minimum or maximum override value	The override controller vibrates as soon as the minimum or maximum override value is reached.
Override value of 100%	The override controller vibrates as soon as the override value is at 100%.
Stop at the breakpoint	The override controller vibrates as soon as the control stops at a breakpoint.

The Program run options window
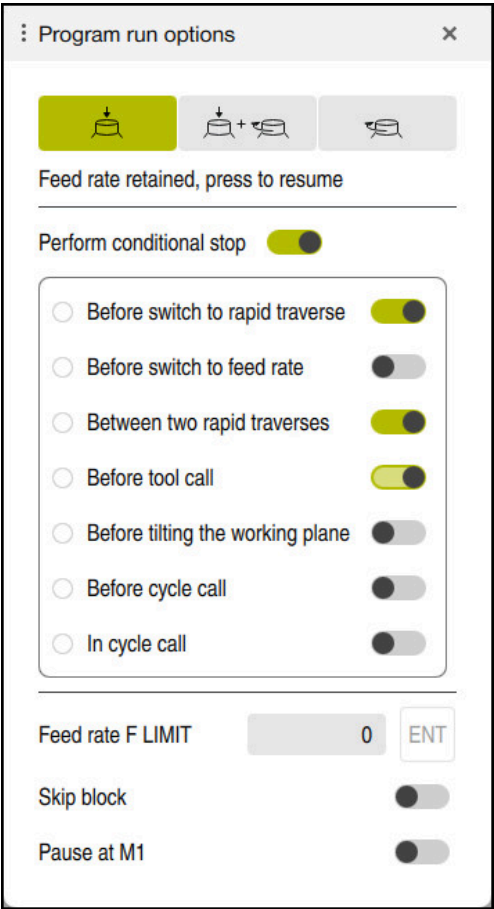
The **Program run options** window

You can open the **Program run options** window as follows:

- In the **Program Run** operating mode with the **Program run options** button
  Further information: "Icons and buttons", Page 1640
- In the **Simulation** workspace with the **Program run options** toggle switch in the **Visualization options** column
  Further information: "The Visualization options column", Page 1200

The following settings of the **Program run options** window are relevant for the override controller:

Icon or button	Meaning
	Feed rate retained, press to resume When this button is active, the control does not change the override value when stopping at a breakpoint. Continue the NC program by pushing the **NC Start** button.
	Feed rate set to 0%, press and turn to resume When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by pushing the **NC Start** button and increasing the override value.
	Feed rate set to 0%, turn to resume When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by increasing the override value.

Icon or button	Meaning
	⚙ Refer to your machine manual. The machine manufacturer uses the optional machine parameter **resumeByTurning** (no. 141801) to define if this button is available.
Perform conditional stop	Toggle switch for activating and deactivating breakpoints **Further information:** "Breakpoints", Page 1761

ℹ The following functions are available also without the override controller:
- **Feed rate F LIMIT**
  Further information: "Feed rate limit F LIMIT", Page 1642
- **Skip block**
  Further information: "Hiding NC blocks", Page 1165
- **Pause at M1**
  Further information: "Overview of miscellaneous functions", Page 965

Breakpoints

The control offers the following breakpoints:

Breakpoint	Meaning
Before switch to rapid traverse	The control stops at each change from the feed rate **F** to rapid traverse **FMAX**.
Before switch to feed rate	The control stops at each change from rapid traverse **FMAX** to the feed rate **F**.
Between two rapid traverses	The control stops between two directly sequential **FMAX** rapid traverse movements.
Before tool call	The control stops before every physical tool call with **TOOL CALL**. ℹ The control does not stop, for example, before a **TOOL CALL** that simply changes the spindle speed.
Before tilting the working plane	The control stops before NC blocks with the following syntax elements: ■ **PLANE** functions (#8 / #1-01-1) ■ **M128** (#9 / #4-01-1) ■ **FUNCTION TCPM** (#9 / #4-01-1) ■ Cycle **19 WORKING PLANE** (#8 / #1-01-1) ℹ You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.
Before cycle call	The control stops before NC blocks with the following syntax elements: ■ **M89** The control stops before each machining position. ■ **M99** ■ **CYCL CALL** ■ **CYCL CALL POS** ■ **CYCL CALL PAT** The control stops before each machining position. ■ Cycles **220 POLAR PATTERN**, **221 CARTESIAN PATTERN**, **224 DATAMATRIX CODE PATTERN** The control stops before each machining position.
In cycle call	**Stop before the first infeed** In the cycles below, the control stops before the first infeed: ■ Cycles for drilling and thread machining **Further information:** "Cycles for Drilling, Centering and Thread Machining", Page 445 ■ Cycle **255 ENGRAVING** **Further information:** "Cycle 225 ENGRAVING ", Page 681 **Stop before every infeed** In the cycles below, the control stops before every infeed: ■ Milling Cycles **Further information:** "Milling Cycles", Page 535 **No stop**

Breakpoint	Meaning
	The control will not stop in the following cycles: ■ Programmable touch probe cycles (#17 / #1-05-1) **Further information:** "General information about touch probe cycles", Page 230 ■ Cycle **239 ASCERTAIN THE LOAD** (#143 / #2-22-1) **Further information:** "Cycle 239 ASCERTAIN THE LOAD (#143 / #2-22-1)", Page 913 ■ Cycle **238 MEASURE MACHINE STATUS** (#155 / #5-02-1) **Further information:** "Cycle 238 MEASURE MACHINE STATUS (#155 / #5-02-1)", Page 910

The control displays active breakpoints on the **PGM** tab of the **Status** workspace.

Further information: "PGM tab", Page 163

Displaying breakpoints

The control displays breakpoints with the following icons:

Icon	Meaning
●	Active stop The control has detected a breakpoint and stops program run or the simulation at this point.
●	Inactive stop The control has detected a breakpoint but does not stop program run or the simulation at this point. In order to stop before this NC block, you must first activate the corresponding toggle switch in the **Program run options** window. **Further information:** "The Program run options window", Page 1759

The control displays the icons for breakpoints in the NC program before the block number as soon as at least one conditional stop is active in the **Program run options** window.

When you select an icon, the control displays the name of the associated breakpoint.

Notes

- The override controller is also effective as a feed rate and/or rapid traverse override in the **Manual** operating mode.
- If the NC program contains breakpoints, the control displays a check mark in the **Perform conditional stop** area of the **Tool check** column.

  Further information: "The Tool check column in the Program workspace", Page 288
- If you turn the override controller down with a sudden jerk, the control will automatically set the override value to 0%, even if the override controller has not reached that position.
- When the execution cursor reaches a breakpoint, the two icons overlap so you can see why the control stops.
- If the **Feed rate set to 0%, turn to resume** button is active, the control reacts as follows:
    - You can continue the NC program only following a conditional stop and by increasing the override value. Otherwise an **NC Start** is necessary (e.g., when starting a program).
    - When the NC program includes two subsequent conditional stops, the 0% override value cannot be changed for 0.3 seconds. This way, the control ensures that you will not continue beyond both conditional stops by just one movement of the Override Controller.
    - After a conditional stop with a manual tool change you must press the **NC Start** button. You can't continue the NC program by increasing the override value.

Notes about machine parameters

Refer to your machine manual.

- The machine manufacturer defines the maximum override value for rapid traverse. If the maximum override value is, for example, 100% and you enter a rapid-traverse override value greater than 100%, the control still calculates with 100%. If you turn the dial down in this case, then there is no immediate effect. Only once the override controller actually reaches 100% will the control change the override value.
- The machine manufacturer can use the optional machine parameter **ocWaitTime** (no. 103412) to define whether a waiting time will be effective in the cases below:
    - When the program is continued at 0% after a breakpoint
    - When 100% of the override value is reached

41

Embedded
Workspace
and Extended
Workspace

41.1 Embedded Workspace (#133 / #3-01-1)

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 to connect the Windows PC (#133 / #3-01-1).

Related topics

■ Remote Desktop Manager (#133 / #3-01-1)

Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

■ Using Extended Workspace to operate a Windows PC through an additional connected monitor

Further information: "Extended Workspace", Page 1768

Requirements

■ Established RemoteFX connection to the Windows PC through Remote Desktop Manager (#133 / #3-01-1)

■ 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 1820

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 94
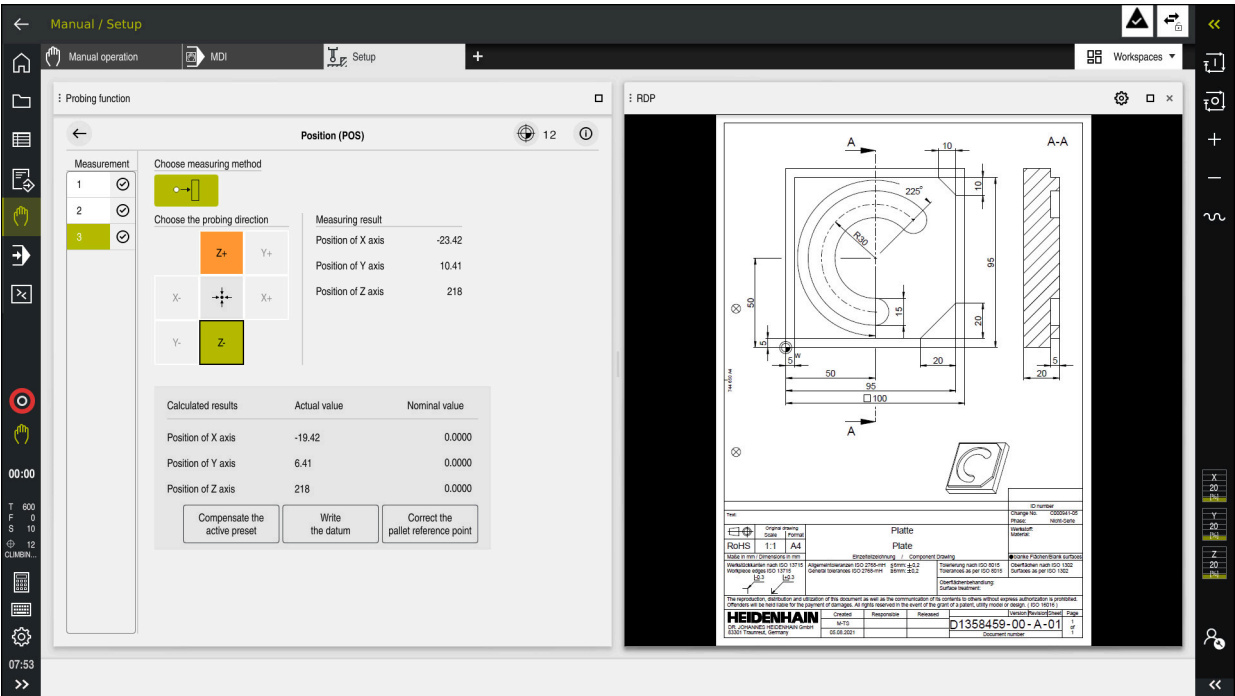
Embedded Workspace as workspace with opened PDF file

The 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.

41.2   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 (#133 / #3-01-1)

  Further information: "Embedded Workspace (#133 / #3-01-1)", Page 1766

■ ITC hardware expansion

  Further information: "Hardware enhancements", Page 89

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 1868

■ Displaying and operating computers connected through Remote Desktop Manager (#133 / #3-01-1)

  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

42

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

- On controls with **SIK1**:
  - Integrated functional safety (FS, basic version; software option 160) or Integrated functional safety (FS, full version; software option 161)
  - 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.
- On controls with **SIK2**:
  - Software option FS, basic version (#6-30-1)
  - Software option FS, Safe axes (#6-30-2), if applicable
    If your control is equipped with **SIK2**, software option #6-30-1 will enable four safe axes. You can order software option #6-30-2* multiple times and thus enable up to six additional safe axes.
- 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: "The Axis status menu item", Page 1773

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: Setup mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified users
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 monitored operating states of the speed **S** and feed rate **F** are displayed 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).
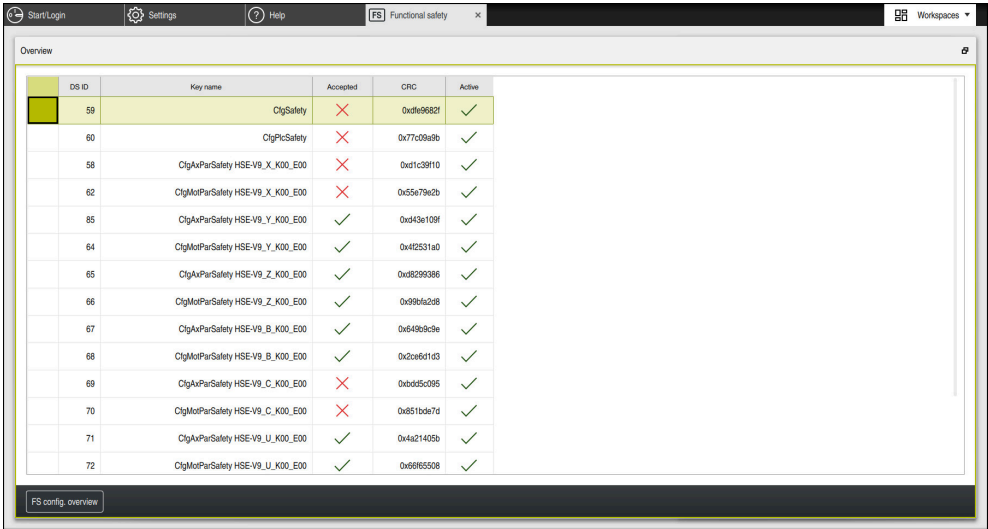
Further information: "Axis display and position display", Page 150

The 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.

The **Overview** workspace in the **Functional safety** application

The 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 1772
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**.

The **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 1775

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 1775
⚠	The axis is not monitored by functional safety (FS) or is not configured as a safe axis. The axis is monitored by functional safety (FS), but the **SLP** safety function is deactivated. In machine parameter **safeAbsPosition** (no. 403130), the machine manufacturer defines whether the **SLP** safety function is activated for an axis.

Feed-rate limiting with functional safety (FS)

> ⚙ Refer to your machine manual.
> This function must be adapted by your machine manufacturer.

With the **F limited** toggle switch you can prevent the SS1 reaction for safe stopping of drives when the guard door is opened.

With the **F limited** toggle switch the control limits the speed of the axes and rotational speed of the spindle 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.

> ℹ In the safety-related operating mode SOM_1, the control stops the axes and spindles when the guard door is opened.

In the **Positions** and **Status** workspaces, the feed rate is displayed in orange.

Further information: "POS tab", Page 164

42.1 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).

43

The Settings Application

43.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Icon	Menu item
🖵	**Machine Settings**	🖵	**Machine Settings** **Further information:** "The Machine Settings menu item", Page 1781
		ⓘ	**General Information** **Further information:** "The General Information menu item", Page 1784
		⬚	**SIK** **Further information:** "The SIK menu item", Page 1785
		🕐	**Machine Times** **Further information:** "The Machine Times menu item", Page 1788
		⚒	**Set Up Touch Probes** **Further information:** "Setting up touch probes", Page 1228
		⚒	**Set Up Wireless Handwheel** **Further information:** "HR 550FS wireless handwheel", Page 1751
OS	**Operating System**	🗓	**Date/Time** **Further information:** "The Adjust system time window", Page 1789
		A文	**Language/Keyboards** **Further information:** "Conversational language of the control", Page 1790
		OSⓘ	**About HeROS** **Further information:** "Information on licensing and use", Page 84
		⊗	**SELinux** **Further information:** "SELinux security software", Page 1791
		👥	**UserAdmin** **Further information:** "The User administration window", Page 1850
		👤	**Current User** **Further information:** "The Active user window", Page 1850
		👆	**Touchscreen Configuration** You can select the touchscreen sensitivity and define whether touch points should be shown or hidden.
🖧	**Network/Remote Access**	⤳	**Shares** **Further information:** "Network drives on the control", Page 1792

Icon	Category	Icon	Menu item
		🔲	**Network** **Further information:** "Ethernet interface", Page 1795
		📋	**PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "PKI Admin", Page 1802
		⇄	**OPC UA** **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804
		⇄	DNC **Further information:** "The DNC menu item", Page 1810
		⊠	**Embedded Workspace** Show the connection status **Further information:** "Embedded Workspace (#133 / #3-01-1)", Page 1766
		🖨	**Printer** **Further information:** "Printers", Page 1812
		VNC	**VNC** **Further information:** "The VNC menu item", Page 1815
		⊠	**Remote Desktop Manager** **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819
		VNC	**Real VNC Viewer** Define settings for external software accessing the control (e.g., for maintenance purposes); for network specialists
		🛡	**Firewall** **Further information:** "Firewall", Page 1825
🔍	**Diagnostics/Maintenance**	>_	**Terminal program** Enter and execute console commands
		📋	**HeLogging** Define settings for internal diagnostic files
		🔍	**Portscan** **Further information:** "Portscan", Page 1829
		∿	**perf2** Check processor load and process load
		🗄	**NC/PLC Restore** **Further information:** "Backup and restore", Page 1829
		🔲	**TNCdiag** **Further information:** "TNCdiag", Page 1832

Icon	Category	Icon	Menu item
		⩕⩗	**TNCscope** Software for data recording
		🗄	**NC/PLC Backup** **Further information:** "Backup and restore", Page 1829
		⊡	**Touchscreen Cleaning** The control disables the touchscreen for input for 90 seconds.
		↻	**Update the documentation** **Further information:** "Update the documentation", Page 1832
🏭	**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 1833
🔧	**Configurations**	🔧	**Configurations** **Further information:** "Configuring the control's user interface", Page 1838
FS	**Functional safety**	FS	**Axis status** **Further information:** "The Axis status menu item", Page 1773
		FS	**Safety parameters** **Further information:** "The Functional safety application", Page 1772

43.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field. This input field is accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Meaning
123	Editing machine-specific user parameters
	Further information: "Machine parameters", Page 1833
555343	Special functions for programming with variables
	Further information: "Variable Programming", Page 1005
	Special functions defining the machine behavior
	Further information: "Special functions defining the machine behavior", Page 1971
0	Resetting active code numbers

> **i** The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

43.3 The 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: "The Simulation settings window", Page 1204

Description of function

To navigate to this function:
Settings ▶ Machine Settings ▶ Machine Settings

The 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 before entering it.

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, have been 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 287
	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 957

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 1702
- If the control generated a tool-usage file for an NC program, the **T usage order** and **Tooling list** tables contain data (#93 / #2-03-1).
  Further information: "T usage order (#93 / #2-03-1)", Page 1704
  Further information: "Tooling list (#93 / #2-03-1)", Page 1706

43.4 The General Information menu item

Application

In the **General Information** menu item of the **Settings** application, the control provides information about the control and the machine.

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ General Information

The 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).

The 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 1806

The Machine information area

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 1806

43.5 The 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 78

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ SIK

The SIK Information area

The control displays the following information:

- **Serial Number**
- **ID number**
- **Control Model**
- **Performance Class**
- **Features**
- **Status**
- **Temporarily enable options / Disable options**

The Machine manufacturer key area

In the **Machine manufacturer key** area, the machine manufacturer can define a manufacturer-specific password for the control.

The 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.

The 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
	On controls with **SIK2**, the part number and the name of the software option are displayed.
	The control indicates the status of the software option by means of the following symbols:
	■ No symbol: The software option is not enabled.
	■ Checkmark: The software option is enabled permanently with all functions.
	■ Clock symbol: The software option has been enabled for a limited period of time or can be ordered again on controls with **SIK2**.
	■ Padlock: The software option has been locked by the machine manufacturer.
Expiration Date or **Status**	The control displays the following information on the status of the software option:
	■ **Enabled**
	■ **YYYY-MM-DD**
	If a software option has been enabled for a limited period of time, the control shows the date up to which it will be available.
	■ **X of X**
	On controls with **SIK2**, the control shows how often the software option has been enabled.
Details	Detailed information for the machine manufacturer
Config.	Function that the machine manufacturer can use to lock software options

43.5.1 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**.

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**.
	The software options have been saved on the **SIK**. The TNC7 basic can be equipped with a **SIK1** or **SIK2** plug-in board. Depending which one is used, the numbers of the software options differ.

43.6 The Machine Times menu item

Application

In the **Machine Times** menu item of the **Settings** application, the control shows the run times since commissioning.

Related topics

- Date and time of the control
  Further information: "The Adjust system time window", Page 1789

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **Machine Times**

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.

43.7 The 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 NTP server synchronization.

Related topics

■ Run times of the machine tool
Further information: "The Machine Times menu item", Page 1788

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Date/Time

The **Adjust system time** window contains 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.

43.8 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 1833

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Language/Keyboards

You can't define two different conversational languages for the operating system and control.

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

43.8.1 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

Save	▶ Select **Save** > The control opens the **Configuration data changed. All changes.** window.
Save	▶ Select **Save** > The control opens the notification menu and displays a "Question type" error.
CLOSE CONTROL	▶ 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 1791

■ **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

43.9 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 (such as 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 1825

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ SELinux

By default, **SELinux** access control is implemented as follows:

■ The control executes only programs that are installed with the HEIDENHAIN NC software.

■ Safety-relevant files, such as **SELinux** system files or HEROS boot files, may only be modified using explicitly selected programs.

■ 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** in the **Security Policy Configuration** window is usually protected by a password defined by the machine manufacturer. Please refer to the 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.

43.10 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 822

■ Network settings

Further information: "Ethernet interface", Page 1795

Requirements

■ Existing network connection

■ Control and computer in same network

■ Path and access data of drive to be connected are known

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **Shares**

You can define any number of network drives, but only seven can be connected at a time.

The 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 1794
Remove	Delete an existing connection
Copy	Copy connection
	Further information: "Mount assistant window", Page 1794
Edit	Edit the connection settings
	Further information: "Mount assistant window", Page 1794
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.

The 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 names must be all uppercase letters, terminated by a colon (**:**). ▪ **Private network drive** With user administration 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** With user administration 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 1795
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 include the domain in the user name, but rather specify it as an option.
vers=3.1.1	Protocol version
sec=ntlmssp	Authentication method ntlm
	Use this option if the control displays the **Permission denied** error message upon connecting.

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
nfsvers=2	Protocol version

> **i** If you use the CIMCO NFS software, you must set this option. CIMCO NFS supports NFS only up to version 2.

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

43.11 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 1825
- Network drives on the control
  Further information: "Network drives on the control", Page 1792
- External access
  Further information: "The DNC menu item", Page 1810

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

10BaseT / 100BaseTx / 1000BaseTx
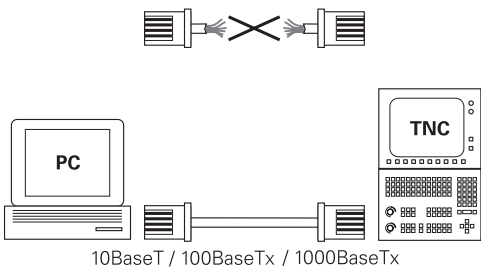
You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

Use a Twisted Pair cable to connect the control to your network.

> ℹ 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 1872 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.

43.11.1 The Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

> ℹ️ Have a network specialist configure the control.

Related topics

- Network configuration
  Further information: "Network configuration with Advanced Network Configuration", Page 1883
- Firewall settings
  Further information: "Firewall", Page 1825
- Network drives on the control
  Further information: "Network drives on the control", Page 1792

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Network

The **Network settings** window

The Status tab

The **Status** tab contains the following information and settings:

Domain	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name.
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the **address** and the **port** of a proxy server in the network.
Interfaces	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:** "The Interfaces tab", Page 1799

DHCP client	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 IP 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.

The 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 toggle 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 1883 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 1883

> ℹ ■ 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.

The 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 and, if necessary, 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.

The 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.

> **ℹ** **Further information:** Programming station for milling controls
> You download the documentation together with the programming station software.

Exporting and importing a network profile

To export a network profile:
- ▶ Opening the **Network settings** window
- ▶ Select **Export configuration**
- ▷ The control opens a window.
- ▶ Select the storage location for the network profile (e.g., **TNC:/etc/sysconfig/net**)
- ▶ Select **Open**
- ▶ Select the desired network profile
- ▶ Select **Export**
- ▷ The control saves the network profile.

> 🛈 You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:
- ▶ Open the **Network settings** window
- ▶ Select **Import the configuration**
- ▷ The control opens a window.
- ▶ Select the storage location of the network profile
- ▶ Select **Open**
- ▶ Select the desired network profile
- ▶ Press **OK**
- ▷ The control opens a window with a prompt.
- ▶ Press **OK**
- ▷ The control imports and activates the selected network profile.
- ▶ You might need to restart the control

> 🛈 Use the **HEIDENHAIN presettings** button to import the default values of the network settings.

Notes

- ■ Preferably restart the control after making changes in the network settings.
- ■ The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.
  Further information: "Conversational language of the control", Page 1790

43.12 PKI Admin

Application

With **PKI Admin**, you can manage the server and client certificates on the control. To define access rights to the control, you can classify the certificates as trusted or not trusted, for example.

Related topics

- Quickly and easily connecting the OPC UA client application to the control (#56-61 / #3-02-1*)

   Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 1808

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **PKI Admin**

The **Administration of the PKI Infrastructure** window contains the following tabs:

Tab	Function
Trusted	The server knows the certificate and trusts it after successful validation.
	For connection to the server, the client certificate must have been specified on this tab.
	For a **OPC UA** connection (#56-61 / #3-02-1*), you also need to assign a **OPC UA** license to the certificate.
	Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 1809
Issuers	On this tab, you can specify the issuer of the trusted certificates.
	The server uses the issuer's information to validate the certificate.
Rejected	On this tab, the control specifies client certificates whose connection attempt to the **OPC UA NC Server** (#56-61 / #3-02-1*) failed.
	Connection failures can occur in the following situations:
	■ The client certificate is unknown and has not been classified as trusted.
	If you want to connect the client application to the server, you can use the **Move** function to move the certificate to the **Trusted** tab.
	■ A trusted client certificate has expired.
Revocation lists	On this tab, you can specify CRL files that list untrusted certificates.
	The server prohibits connections that use these certificates.
Own certificates	The control provides the following functions:
	■ **Recreate certificate**
	The control recreates the server's chain of trust. After the next restart of the control, it will use the new certificate.
	■ **Export certificate chain**
	The control saves the server's chain of trust that you import into the client application.
	■ **Load certificate**
	You can import a customized certificate.
	Please note the requirements for self-created certificates for **OPC UA** (#56-61 / #3-02-1*).
	Further information: "Required certificates", Page 1806
	■ **Check the configuration**
	The control checks the validity of the server certificates.
Advanced settings	The tab contains the following areas:
	■ **Certificate settings**

Tab	Function
	The control adds static IP addresses to the server certificates. You can select the IP address of the **eth0** or **eth1** interface or specify the required IP addresses.
	■ **Settings for revocation lists**
	You can permit connections of applications with certificates in a multi-level certificate chain even if no associated CRL files exist.

Definition

PKI

PKI (public key infrastructure) is the management structure for digital certificates that are required for safe communication. A digital certificate has the same purpose as an identity card or passport. With a digital certificate, its owner can encrypt, sign and authenticate the communication.

43.13 OPC UA NC Server (#56-61 / #3-02-1*)

43.13.1 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 security analysis was updated in 2022. 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.

> 📖 Pay attention to the interface documentation of the **OPC UA NC Server** as well as the documentation of the client application.

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**

■ Quickly and easily connecting the OPC UA client application to the control

Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 1808

Requirements

■ OPC UA NC Server software options (#56-61 / #3-02-1*)

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).

If your control features a **SIK2**, you can order this software option multiple times and enable up to six connections.

■ Firewall configured

Further information: "Firewall", Page 1825

■ The OPC UA client supports the **security policy** and authentication method of the **OPC UA NC Server**:

■ **Security Mode: SignAndEncrypt**

■ **Algorithm:**

■ **Basic256Sha256**

■ **Aes128Sha256RsaOaep**

■ **Aes256Sha256RsaPss**

■ **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

■ Creation of service files

■ Read and write tool data (the corresponding right is required)

■ File system access to the **TNC:** drive

■ File system access to the **PLC:** drive (the corresponding right is required)

■ Validation of 3D models for tool carriers

Further information: "Tool carrier management", Page 273

■ Validate 3D models for tools (#140 / #5-03-2)

Further information: "Tool model (#140 / #5-03-2)", Page 277

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: "The Machine information area", Page 1784
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)
  Further information: "The Info about machine manufacturer area", Page 1784

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. As long as the operating system is active, you can create and transmit service files, for example.

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.

Further information: "PKI Admin", Page 1802

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**.

43.13.2 The OPC UA (#56-61 / #3-02-1*) menu item

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 NC Server**.

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **OPC UA**

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether the **OPC UA NC Server** is active: ■ Green icon **OPC UA NC Server** is active ■ Gray icon: **OPC UA NC Server** is not active or software option not enabled You can manually start or restart the **OPC UA NC Server** as required. **Further information:** "Manually starting the OPC UA NC Server", Page 1808
OPC UA connection assistant	Open the **OPC UA NC Server connection assistant** window **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 1808
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 1809
PKI Admin	Open the **Administration of the PKI Infrastructure** window **Further information:** "PKI Admin", Page 1802
Host computer operation	Activate or deactivate host computer operation with a toggle switch **Further information:** "The DNC area", Page 1811

Manually starting the OPC UA NC Server

You can manually start or restart the **OPC UA NC Server** as required. Thus, you can apply changes made to the machine parameters or the certificates, which are relevant to the server, without having to shut down the control.

While an OPC UA connection is active, the control displays a confirmation prompt before the restart. During the restart, the control will disconnect active connections automatically.

For this function, you need the HEROS.SetNetwork permission.

Further information: "User administration roles and rights", Page 1966

43.13.3 The OPC UA connection assistant function (#56-61 / #3-02-1*)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection assistant** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

■ Assigning the OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-6 using the **OPC UA NC Server - License Settings** window
  Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 1809
■ Managing certificates with the **PKI Admin** menu
  Further information: "PKI Admin", Page 1802

Description of function

Use the **OPC UA** menu item to open the **OPC UA NC Server connection assistant** window.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 1807

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 software option is active for the OPC UA NC Server, the control will generate the server certificate as a part of a self-generated certificate chain during the first start-up. 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.

Note

The **OPC UA NC Server connection assistant** 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.

43.13.4 The OPC UA license settings function (#56-61 / #3-02-1*)

Application

You can use the **OPC UA NC Server - License Settings** window to assign an OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-6.

Related topics

- Setting up the OPC UA client application with the **OPC UA connection assistant** function

  Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 1808
- Managing certificates with **PKI Admin**

  Further information: "PKI Admin", Page 1802

Requirement

- Certificate has been added to the **Trusted** category in **PKI Admin**

Description of function

Use the **OPC UA** menu item to open the **OPC UA license settings** window.

After using the **OPC UA connection assistant** 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 enable the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

43.14 The 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 1792
- Setting up a network
  Further information: "Ethernet interface", Page 1795
- TNCremo
  Further information: "PC software for data transfer", Page 1875
- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ DNC

The **DNC** area contains the following symbols:

Icon	Meaning
✚	**Add** a computer-specific connection
✎	**Edit** a computer-specific connection
🗑	**Delete** a computer-specific connection

When a connection is active, the control displays a symbol in the information bar:

Icon	Meaning
⇄🔒	**Secure connection configuration** External access to the control is active; all connections are using a secure connection configuration.
⇄🔓	**Non-secure connection configuration** An external access to the control is active but at least one connection is using a non-secure connection configuration.

Further information: "Areas of the control's user interface", Page 91

The DNC area

In the **DNC** area you use toggle 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.

Secure connections for user

In the **Secure connections for user** area you activate the following functions:

Row	Meaning
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user.
Key management	In this row you, open the **Certificate and keys** window. **Further information:** "SSH-secured DNC connection", Page 1863

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** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

43.15 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 1028

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 1815

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Printer ▶ Heros Printer Manager

You can print the following files:

■ Text files

■ Graphic files

■ PDF files

Further information: "File types", Page 829

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 1815

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	Selects a printer
DOWN	
STATUS	Displays the status information of the selected printer
PRINT A TEST PAGE	Prints a test page on the selected printer

The Change the printer window
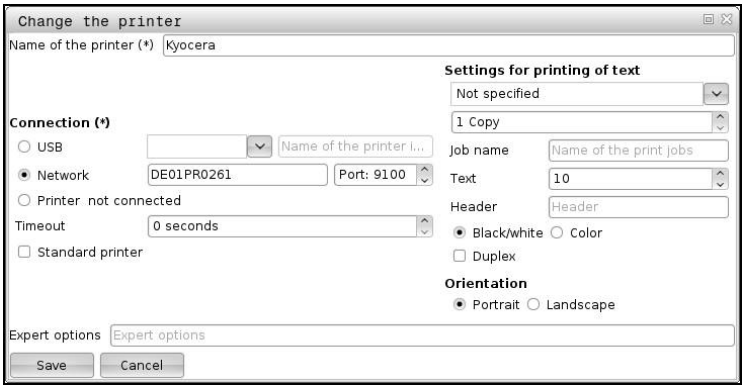
For each printer, the following properties can be set:

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

43.15.1 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.

43.16 The 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 1825
- ■ Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ VNC

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	Enable 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.

The 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.

The Global settings area

In the **Global settings** area, you can define the following settings:

Function	Meaning
Enable RemoteAccess 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.

The 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.

The VNC Focus Settings area

In the **VNC Focus Settings** area, you can define 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 if the focus changes.
	Only if the **Enabling VNC focus** check box is enabled
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 enabled
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 enabled

> **i** Enable 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.

43.17 The Remote Desktop Manager window (#133 / #3-01-1)

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: "The DNC menu item", Page 1810

Requirements

■ Software option Remote Desktop Manager (#133 / #3-01-1)
■ Existing network connection
   Further information: "Ethernet interface", Page 1795

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Remote Desktop Manager

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 1820
■ **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control
   Further information: "VNC", Page 1820
■ **Switch-off/restart of a computer**: Automatically shut down a Windows computer together with the control
■ **World Wide Web**: Available only to authorized specialists
■ **SSH**: Available only to authorized specialists
■ **XDMCP**: Available only to authorized specialists
■ **User-defined connection**: Available only to authorized specialists
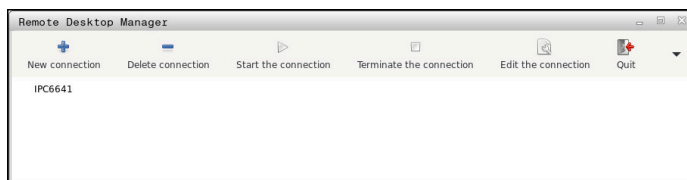
HEIDENHAIN offers the IPC 6641 as a Windows computer. With the IPC 6641 you can start and operate Windows-based applications directly from within the control.

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	Create a new connection in the **Edit the connection** window
	Further information: "Establishing and starting a connection", Page 1823
Delete connection	Delete the selected connection
Start the connection	Start the selected connection
	Further information: "Establishing and starting a connection", Page 1823
Terminate the connection	Terminate the selected connection
Edit the connection	Edit the selected connection in the **Edit the connection** window
	Further information: "Connection settings", Page 1821
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection
	Further information: "Exporting and importing connections", Page 1824
Export the connections	Back-up the selected connection
	Further information: "Exporting and importing connections", Page 1824

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 1823

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	Usage
Connection name	Name of the connection in **Remote Desktop Manager**	Required
	ℹ 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 _	
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	Usage
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	Usage
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	Usage
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.	Required
	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.	
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible.	Required
	Windows applications may delay the shutdown of the computer after port 445 has been closed.	
Force	Close all programs on the Windows computer, even if dialogs are still open.	Required
	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.	
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

43.17.1 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 connections to this computer**

▶ Press **OK** to confirm your settings

43.17.2 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 1821

▶ Press **OK**

> The control saves the settings and closes the window.

▶ Select connection

▶ Select **Start the connection**

> The control starts the connection.

43.17.3 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.

43.18 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 1795
- SELinux security software
  Further information: "SELinux security software", Page 1791

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Firewall

If you activate the firewall, the **Firewall settings** window displays a symbol at the bottom right of the taskbar. The control displays the following symbols, 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 interface configuration, but the DHCP server has not yet assigned an IP address.
	Further information: "The DHCP server tab", Page 1799
🛡️🛡️	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 inhibited 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.
Service	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)

Setting	Meaning
	ⓘ The control might not support connection configurations that use the LSV2 protocol. When the control detects a non-secure connection, it displays a warning message with additional information. In this case, please contact the provider of the corresponding application. HEIDENHAIN recommends the use of the OPC UA or DNC application for access to the control. **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804 **Further information:** "The DNC menu item", Page 1810
	■ **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:** "The VNC menu item", Page 1815
Method	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**.
Log	The control shows the following messages when transmitting network packets: ■ Red: Network packet blocked ■ Blue: Network packet accepted
Computer	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 for the **Permit some** method
Advanced options	Only for network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

Notes

- Have your network specialist check and, if necessary, change the standard settings.
- 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 authentication.
- 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.

43.19 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 1825

■ Network settings

Further information: "Network configuration with Advanced Network Configuration", Page 1883

Description of function

To navigate to this function:

Settings ▶ Diagnostics/Maintenance ▶ Portscan

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)

Start the portscan manually by selecting the **Start** button in the **Manual Execution** area. In the **Automatic Execution** area, you can use the **Automatic update on** function to specify that the control will perform the portscan automatically in the selected 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.

43.20 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 822

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **NC/PLC Backup**

Settings ▶ **Diagnostics/Maintenance** ▶ **NC/PLC Restore**

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 double-click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: "File management", Page 822

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.

43.20.1 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.

43.20.2 Restoring data

<table>
<tr><td colspan="1" align="center">***NOTICE***</td></tr>
<tr><td>**Caution: Data may be lost!**</td></tr>
<tr><td>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</td></tr>
</table>

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.

43.21 TNCdiag

Application

The control displays status and diagnostic information of HEIDENHAIN components in the **TNCdiag** window.

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **TNCdiag**

⚙   Only use this function after consultation with your machine manufacturer.

📖   For more information, please refer to the **TNCdiag** documentation.

43.22 Update the documentation

Application

The **Update the documentation** function can be used, for example, to install or update the integrated **TNCguide** product help.

Related topics

- Integrated product help **TNCguide**

    Further information: "User's Manual as integrated product aid: TNCguide", Page 66

- Product help on the HEIDENHAIN website

    TNCguide

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **Update the documentation**

The file manager is located in the **Update the documentation** area. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 1833

The control shows all available documents in the **Help** application.
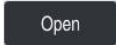
Further information: "The Help workspace", Page 1158

ℹ   In the **Update the documentation** area, you can install all HEIDENHAIN-specific documents (e.g., NC error messages).

43.22.1 Transferring TNCguide

You can find and transfer the desired **TNCguide** version as follows:

▶ Select the link to the HEIDENHAIN website
https://content.heidenhain.de/doku/tnc_guide/html/de/index.html

▶ Select **TNC Controls**

▶ Select **TNC7 Series**

▶ Select the NC software number

▶ Navigate to **Product help (HTML files)**

▶ Select **TNCguide** in the desired language

▶ Select path to save the file

▶ Select **store**

> The download begins.

▶ Transfer the downloaded file to the TNC control

⌂
    ▶ Select the **Home** operating mode

    ▶ Select the **Settings** application

    ▶ Select **Diagnostics/Maintenance**

    ▶ Select **Update documentation**

    > The control opens the **Select installation file** area.

    ▶ Select the desired file with extension ***.tncdoc**

Open
    ▶ Select **Open**

    > A pop-up window appears, stating whether the installation was successful or failed.

    ▶ Select the **Help** application

⌂
    ▶ Select **home**

    > The control shows all available documentation.

43.23 Machine parameters

Application

You can configure the behavior of the control with machine parameters. For this purpose, the control provides the **MPs for Users** and **MPs for setters** applications. 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 for the **MPs for setters** application

Further information: "Machine parameters", Page 1890

Requirements

■ Code number 123

Further information: "Code numbers", Page 1781

■ The contents of the **MPs for setters** application have been defined by the machine manufacturer

Description of function

To navigate to this function:

Settings ▶ Machine Parameters ▶ MPs for setters

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:

- **Details**
- **Document**
- **List**

You cannot close the **List** workspace.

Configuration editor areas
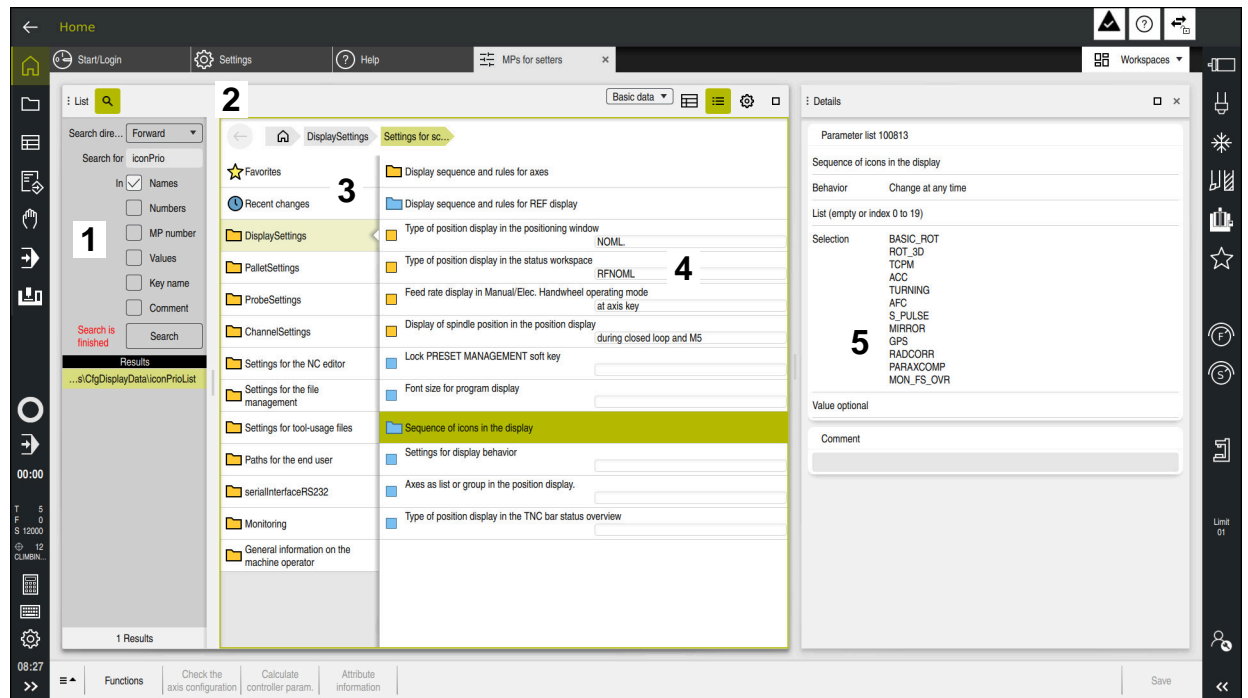
The **MPs for setters** application with a machine parameter selected

The configuration editor shows the following areas:

1 The **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

The title bar of the **List** workspace includes the following functions:

- Open or close the **Search** column

- Filter contents using a selection menu

- Toggle between structure and table views

In the table view, you can compare data objects.

The control displays the following information:

- Name of the objects

- Symbols of the objects

- Machine parameter values

■ Open or close the **Details** workspace
Further information: "The Details workspace", Page 1838

■ Open or close the **Configuration** window
Further information: "Configuration window", Page 1838

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 The **Details** workspace

The control displays information on the selected machine parameter or the most recent change you made.
Further information: "The Details workspace", Page 1838

Icons and buttons

The configuration editor contains the following icons and buttons:

Icon or button	Meaning
⊞	Activate or deactivate the **table view**
	The control toggles between structure and table views.
	Further information: "Configuration editor areas", Page 1835
☰	Open or close the **Details** workspace
	Further information: "The Details workspace", Page 1838
⚙	Open or close the **Configuration** window
	Further information: "Configuration window", Page 1838
🕐	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 not readable and not editable
✏	Changes to the machine parameter not yet saved
Functions	Open the context menu
	Further information: "Context menu", Page 1174
Check the axis configuration	Only for the machine manufacturer
Calculate controller param.	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

The **Configuration** window includes the **Show MP descriptive texts** toggle switch.

If the toggle switch is active, the control displays a description of the machine parameter in the active conversational language.

If the toggle switch is not active, the control displays the language-neutral name of the machine parameter.

The Details workspace

If you select contents from the favorites or the structure, the control will display information in the **Details** workspace, such as:

- Type of object, such as data object list or parameter
- Descriptive text of machine parameter
- Permitted or required input
- Prerequisite for the change (e.g., program run blocked)
- Number of the machine parameter on the iTNC 530
- Machine parameter optional

  This information is included if a machine parameter can be enabled optionally.

If you select contents from the most recent changes you made, the control will display the following information in the **Details** workspace:

- Sequential number of the last change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Permitted or required input

43.23.1 Note

The machine manufacturer offers further applications for machine parameters.

If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

43.24 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is individually adapted.

Related topics

- Workspaces

  Further information: "Workspaces", Page 94
- Control interface

  Further information: "Areas of the control's user interface", Page 91

Description of function

To navigate to this function:

Settings ▶ Configurations ▶ Configurations

A configuration contains all adaptations to the control's user interface that do not influence the control's actual functions.

- Settings for the TNC bar
- Arrangement of workspaces
- Font size
- Favorites

The **Configurations** area contains the following functions:

Function	Meaning
Active Configuration	Activate a configuration from a selection menu **Further information:** "The Desktop menu workspace", Page 109
Default configuration	Use the **Reset** button to apply the settings of the **OEM configuration** to the active configuration.
Save as OEM Configuration	The machine manufacturer can use the **Save** button to overwrite the **OEM configuration**.
Save current settings	With the **Save** button, you can save the current version of the active configuration.
Restore last configuration	With the **Reset** button, you can discard any customizations and revert to the saved version of the active configuration.

The control displays the following information about all available configurations in a table:

Column	Meaning
Configuration Name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 1840
Edit	This column contains two buttons, for renaming and deleting the configuration.

Press the **Add** button to create a new configuration.

43.24.1 Exporting and importing configurations

To export configurations:

▶ Select the **Settings** application

▶ Select **Configurations**

> The control opens the **Configurations** area.

▶ Activate the **Exportable** toggle switch for the desired configuration, if necessary

Export

    ▶ Select **Export**

    > The control opens the **Save as** window.

    ▶ Select the target file

    ▶ Enter a file name

Create

    ▶ Select **Create**

    > The control saves the configuration file.

To import configurations:

Import

    ▶ Select **Import**

    > The control opens the **Import configurations** window.

    ▶ Select file

Import configuration

    ▶ Select **Import**

    > If importing a configuration would overwrite a file with the same name, the control displays a prompt.

    ▶ Select the procedure:

       ■ **Overwrite**: the control overwrites the original configuration.

       ■ **Keep**: the control does not import the configuration.

       ■ **Cancel**: the control cancels the import process.

Notes

■ Delete only inactive configurations. If you delete an active configuration, the control first actives a default configuration. This can lead to delays.

■ The **Overwrite** function permanently replaces existing configurations.

44

User Administration

44.1 Fundamentals

Application

User administration enables you to create and administrate different users with different access rights to various functions of the control. You can assign roles to the various users that reflect their respective tasks, such as machine operator or setup technician.

User administration is inactive in the control's factory default setting. This status is called **Legacy-Mode**.

Description of function

User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:

■ Application security
■ Network security
■ Platform security

The user administration differentiates between the following terms:

■ User
  Further information: "Users", Page 1842
■ Roles
  Further information: "Roles", Page 1844
■ Rights
  Further information: "Rights", Page 1844

User
(for example: Test)
contains several
roles

roles
(for example: NC.Programmer)
contain several rights

rights
(for example: NC.EditNCProgram)
contain several functions
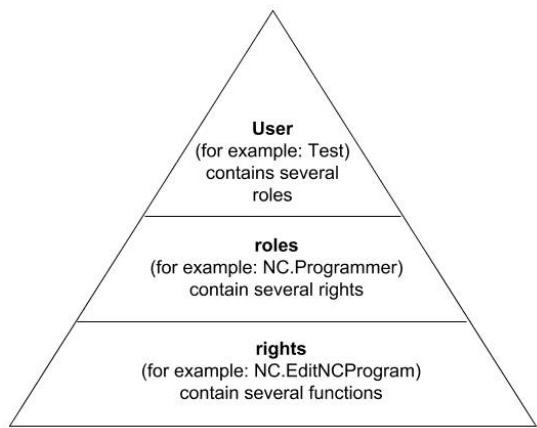
Users

The user administration offers the following types of users:

■ Function users pre-defined by HEIDENHAIN
■ Function users pre-defined by the machine manufacturer
■ Self-defined users

Depending on the task assigned, you can use one of the pre-defined function users or you have to create a new user.

Further information: "Creating a new user", Page 1848

If you deactivate user administration, the control saves all configured users. Thus they will be available again when user administration is reactivated.

If you want to delete the configured users upon deactivation, you need to set this explicitly when deactivating user administration.

Further information: "Deactivating user administration", Page 1849

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.

- **sys**

  The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.

- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.

- **oem**

  The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer defines function users who are required for specific tasks such as machine maintenance.

By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.

Further information: "The Active user window", Page 1850

The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.

Roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

Further information: "List of roles", Page 1966

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

User administration offers roles for the following tasks:

- **Operating system roles:** access to functions of the operating system and interfaces
- **NC operator roles:** access to functions for programming, setting up and running NC programs
- **Machine tool builder (PLC) roles:** access to functions for configuring and checking the control

Every user should have at least one role from the operating system area and at least one role from the programming area.

HEIDENHAIN recommends permitting more than one person to access an account with the HEROS.Admin role. This ensures that necessary changes to user administration can also be made in the administrator's absence.

Local or remote registration

You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.

Further information: "SSH-secured DNC connection", Page 1863

If a role is only enabled for local login, Local. is added to the role name (e.g., Local.HEROS.Admin instead of HEROS.Admin).

If a role is only enabled for remote login, Remote. is added to the role name (e.g., Remote.HEROS.Admin instead of HEROS.Admin).

You can therefore also make the rights of a user dependent on the access used to operate the control.

Rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.

Rights gather various functions of the control (e.g., editing the tool table).

User administration offers rights for the following tasks:

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

> ⓘ Ensure that every user is assigned all access rights he needs. The access rights result from the tasks a user performs on the control.

The access rights of HEIDENHAIN function users are already pre-defined in the control's factory default setting.

Further information: "List of rights", Page 1969

Password settings

If you use an LDAP database, users with the HEROS.Admin role can define password requirements. For this, the control provides the **Password settings** tab.

Further information: "Saving user data", Page 1851

The following parameters are available:

Password lifetime

■ **Validity period of password:**

Here, you can indicate how long the password can be used.

■ **Warning before expiration:**

From the defined time, a warning will be issued that the password will soon expire.

Password quality

■ **Minimum password length:**

Here, you can indicate the minimum password length.

■ **Minimal number of character classes (upper/lower, digits, special):**

Here, you can indicate the minimum number of different character classes required in the password.

■ **Maximum number of repeated characters:**

Here, you can indicate the maximum number of identical successive characters in the password.

■ **Maximum length of character sequences:**

Here, you can indicate the maximum length of the character sequences to be used in the password (e.g., 123).

■ **Dictionary check (number of matching characters):**

Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.

■ **Minimum number of characters changed compared to previous password:**

Here, you can specify how many characters in the new password must be different from the previous one.

You define the values for each parameter on a scale.

For reasons of security, passwords should comply with the following criteria:

■ Eight characters minimum

■ Letters, numbers, and special characters

■ Avoid using whole words or a sequence of characters (e.g., Anna or 123)

> **i** If you want to use special characters, pay attention to the keyboard layout. HEROS assumes a US keyboard, the NC software assumes a HEIDENHAIN keyboard. External keyboards can be freely configured.

Additional directories

HOME: drive

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by the respectively logged-in users as well as users with the HEROS.Admin role.

public directory

Upon the first activation of user administration, the **public** directory below the **TNC:** drive will be connected.

The **public** directory can be accessed by any user.

In the **public** directory you can, for example, make files available to other users.

Further information: "File management", Page 822

44.1.1 Configuring user administration

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

1 Opening the **User administration** window
2 Activating user administration
3 Defining the password for the **useradmin** function user
4 Setting up a database
5 Creating a new user

> 🛈   ■ You can exit the **User administration** window after each configuration step.
>   ■ If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.

Opening the User administration window

To open the **User administration** window:

▶ Select the **Settings** application
▶ Select **Operating System**
▶ Double-tap or double-click **CurrentUser**
> The control opens the **User administration** window in the **Settings** tab.
   Further information: "The User administration window", Page 1850

Activating user administration

To activate user administration:

- ► Select **User administration active**
- > The control shows the message **Password for user 'useradmin' missing**.
- ► Retain or reactivate the active status of the **Anonymize users in log data** function

ⓘ
- ■ The purpose of the **Anonymize users in log data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.
- ■ If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.

Define the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 1842

To define a password for the **useradmin** function user:

- ► Select **Password for useradmin**
- > The control opens the **Password for user 'useradmin'** pop-up window.
- ► Enter the password for the **useradmin** function user

ⓘ Please observe the recommendations for passwords.
Further information: "Password settings", Page 1845

- ► Repeat the password
- ► Select **Set new password**
- > The control shows the message **Settings and password for 'useradmin' were changed**.

Setting up a database

To set up a database:

- ► Select the database for saving your user data (e.g., **Local LDAP database**)
- ► Select **Configuration**
- > The control opens a window for configuring the corresponding database.
- ► Follow the instructions from the control in the window
- ► Select **APPLY**

ⓘ The following options are available for saving your user data:
- ■ **Local LDAP database**
- ■ **LDAP on remote computer**
- ■ **Connection to Windows domain**

Parallel operation of Windows users and users from an LDAP database is possible.
Further information: "Saving user data", Page 1851

Creating a new user

To create a new user:

▶ Select the **User administration** tab
▶ Select **Create new user**
> The control adds a new user to the **User list**.
▶ Change the name as needed
▶ Edit a password as needed
▶ Define a profile image as needed
▶ Enter a description as needed
▶ Select **Add role**
> The control opens the **Add role** window.
▶ Select a role
▶ Select **Add**

> **i**   You can also add roles using the **Add external login** and **Add local login** buttons.
> **Further information:** "Roles", Page 1844

▶ Select **Close**
> The control closes the **Add role** window.
▶ Select **OK**
▶ Select **APPLY**
> The control adopts the changes.
▶ Select **END**
> The control opens the **System reboot required** window.
▶ Select **Yes**
> The control restarts.

> **i**   The user must change the password when logging in for the first time.

44.1.2 Deactivating user administration

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 1842

To deactivate user administration:

▶ Log in as a function user
▶ Opening the **User administration** window
▶ Select **User administration inactive**
▶ If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
▶ Select **APPLY**
▶ Select **END**
> The control opens the **System reboot required** window.
▶ Select **Yes**
> The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in log data** function, the system will show personalized user data in all control log files.
If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case, it is your responsibility to ensure that all required data protection provisions have been made at your company.

▶ Retain or reactivate the active status of the **Anonymize users in log data** function

---

- Some user administration areas are configured by the machine manufacturer. Refer to your machine manual.
- HEIDENHAIN recommends activating user administration as part of an IT safety concept.
- If both user administration and a screensaver are active, then the current user's password must be entered to unlock the screen.
  Further information: "HEROS menu", Page 1868
- If you used **Remote Desktop Manager** to establish private connections before user administration was activated, these connections are no longer available after the activation of user administration. Save your private connections before activating user administration.
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819

44.2 The User administration window

Application

In the **User administration** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- The **Active user** window

  Further information: "The Active user window", Page 1850

Requirement

- If user administration is active, the HEROS.Admin role

  Further information: "List of roles", Page 1966

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ UserAdmin

The **User administration** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration **Further information:** "Configuring user administration", Page 1846
User administration	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 1848
Password settings	Define password requirements **Further information:** "Password settings", Page 1845
User-defined roles	Roles created for a Windows domain **Further information:** "Connection to Windows domain", Page 1854

44.3 The Active user window

Application

In the **Active user** window, the control displays information about the logged on user, such as assigned rights. You can also manage other user settings, such as keys for SSH-secured DNC connections or smartcards for logon, and change the password.

Related topics

- SSH-secured DNC connections

  Further information: "SSH-secured DNC connection", Page 1863
- Logon with smartcards

  Further information: "Logon with smartcards", Page 1861
- Available roles and rights

  Further information: "User administration roles and rights", Page 1966

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Current User

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the **Added rights** tab, enable rights for another user or function user until the next logoff
Open user administration	Open the **User administration** window **Further information:** "The User administration window", Page 1850
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 1863 **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804
Create token	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 1861
Delete token	
Close	Close the **Active user** window

On the **Change password** tab you can check your password against the current requirements or set a new password.

Further information: "Password settings", Page 1845

Note

In legacy mode, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect.

Further information: "Users", Page 1842

44.4 Saving user data

44.4.1 Overview

The following options are available for saving your user data:

- **Local LDAP database**
  Further information: "Local LDAP database", Page 1852
- **LDAP on remote computer**
  Further information: "LDAP database on a remote computer", Page 1853
- **Connection to Windows domain**
  Further information: "Connection to Windows domain", Page 1854

> ⓘ Parallel operation of Windows users and users from an LDAP database is possible.

44.4.2 Local LDAP database

Application

With the **Local LDAP database** setting the control saves the user data locally. That way you can activate user administration even on machines without a network connection.

Related topics

- Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 1853
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 1854

Requirements

- User administration is active
  Further information: "Activating user administration", Page 1847
- **useradmin** user is logged on
  Further information: "Users", Page 1842

Description of function

A local LDAP database offers the following options:

- Using user administration on one single control
- Setting up a central LDAP server for more than one control
- Exporting an LDAP server configuration file if the exported database is to be used by more than one control

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- ▶ Opening the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Configuration**
- > The control opens the **Configure local LDAP database** window.
- ▶ Enter the name of the **LDAP domain**
- ▶ Enter the password
- ▶ Repeat the password
- ▶ Select **OK**
- > The control closes the **Configure local LDAP database** window.

Notes

- Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.
  Passwords must not be trivial and must be known only to the administrators.
- If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

44.4.3 LDAP database on a remote computer

Application

With the **LDAP on remote computer** function you can transmit the configuration of a local LDAP database between controls and computers. That way you can use the same users on multiple controls.

Related topics

- Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 1852
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 1854

Requirements

- User administration is active
  Further information: "Activating user administration", Page 1847
- **useradmin** user is logged on
  Further information: "Users", Page 1842
- LDAP database has been set up in the company network
- Server configuration file of an existing LDAP database is stored on the control or a PC in the network
  If the configuration file is stored on a PC, the PC must be running and accessible through the network.
  Further information: "Providing a server configuration file", Page 1853

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:

- ▶ Opening the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Export server configuration**
- > The control opens the **Export LDAP configuration file window.**
- ▶ Enter the name for the server configuration file into the name field
- ▶ Save the file to the desired folder
- > The control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:

▶ Opening the **User administration** window
▶ Select **LDAP user database**
> The control enables the dimmed area for editing the LDAP user database.
▶ Select **LDAP on remote computer**
▶ Select **Import server configuration**
> The control opens the **Import LDAP configuration file window.**
▶ Select the existing configuration file
▶ Select **FILE**
▶ Select **APPLY**
> The control imports the configuration file.

44.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function, you can connect the data of a domain controller with the control's user administration.

Ask your IT administrator to configure the connection to the Windows domain.

Related topics

■ Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 1852
■ Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 1853

Requirements

■ User administration is active
  Further information: "Activating user administration", Page 1847
■ **useradmin** user is logged on
  Further information: "Users", Page 1842
■ Windows domain controller present in the network
■ Domain controller accessible in the network
■ Organizational unit for HEROS roles known
■ For logon with computer account:
  ■ You have access to the password of the domain controller
  ■ You have access to the user interface of the domain controller or you are supported by an IT administrator
■ For logon with function user:
  ■ User name of the function user
  ■ Password of the function user

Description of function

The control provides the following options to join a Windows domain:

- Create a separate account for the control
- By means of a function user

Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.

Click the **Configuration** button to open the **Configure Windows domain** window.

Further information: "The Configure Windows domain window", Page 1856

The Configure Windows domain window

After the domain search, you can customize the Windows domain information or specify new information in the **Configure Windows domain** window.

Your IT administrator will provide the required information.

The**Configure Windows domain** window contains the following settings:

Setting	Meaning
Domain name:	Server name of the Windows domain
	Is populated by domain search
Key Distribution Center (KDC):	KDC address
	Is populated by domain search
Alternative admin server:	Deviating server name where the passwords are managed
Map SIDs to Unix UIDs	Map the Windows user SIDs (Security IDs) in Active Directory to the matching Unix UIDs on the control
Use LDAPs	Transfer data using secure LDAPs. LDAPs encrypts user data and passwords.
	You can select a certificate or disable certificate validation.
Group for login authorization:	Define a special group of Windows users to whom you want to restrict the connection to this control
Organizational unit for HEROS roles:	Modify the organizational unit in which the HEROS role names are stored
	Specify the configuration of your domain.
Prefix for HEROS role names:	Change the prefix in order to manage users from different workshops, for example. Each prefix given to a HEROS role name can be changed (e.g., HEROS hall 1 and HEROS hall 2)
	Is populated by domain search
Separator for HEROS role names:	Modify the separator within the HEROS role names
Advanced configuration of domain section	Only for IT administrators

If you enable the **Active Directory with function user** check box, the window contains the following additional settings:

Setting	Meaning
Function user:	Enter the user name and password of the Active Directory function user
Organizational unit for function user:	Specify the organizational unit of the function user

The function user's user name must not contain blanks. The name and organizational unit form the complete path (Distinguished Name, DN) in the Active Directory.

Groups of the domain

If not all of the required roles have been created in the domain as groups, the control issues a warning.

If the control issues a warning, proceed in one of the two following ways:

- Use the **Add role definition** function to enter a role directly in the domain
- Use the **Export role definition** function to export the roles to an ***.ldif** file

There are the following ways to create groups corresponding to the different roles:

- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

The Windows administrator must add the users manually to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by below.

Example 1

The user is a direct or indirect member of the respective group:
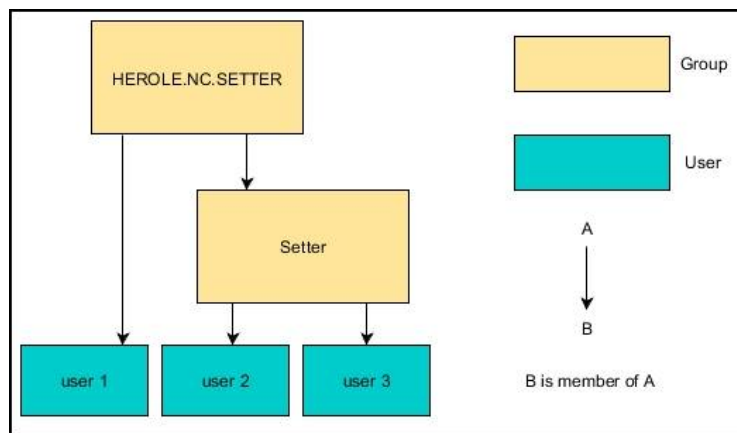
Example 2

Users from various sectors (workshops) are members of groups with different prefixes:
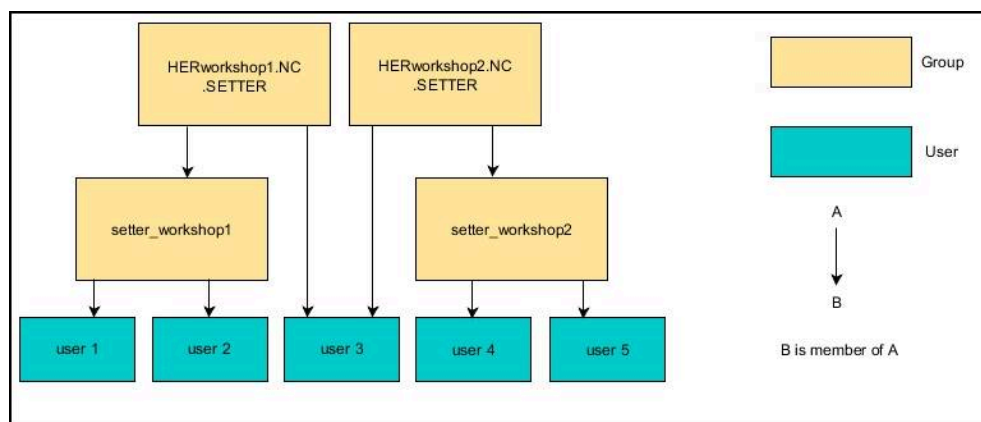
Joining a Windows domain with a computer account

To join a Windows domain with a computer account:

▶ Opening the **User administration** window
▶ Select **Connection to Windows domain**
▶ Select the **Join Active Directory domain (with computer account)** check box
▶ Select **Find domain**
> The control selects a domain.
▶ Select **Configuration**
▶ Check the data for **Domain name:** and **Key Distribution Center (KDC):**
▶ Enter **Organizational unit for HEROS roles:**
▶ Select **OK**
▶ Select **APPLY**
> The control opens the **Connect to domain** window.

⊕   The **Organizational unit for computer account:** function allows you to specify in which of the already existing organizational units you want to create the access, such as
  ◾ ou=controls
  ◾ cn=computers
  The values you enter must match the conditions of the domain. The terms are not exchangeable.

▶ Enter the user name of the domain controller
▶ Enter the password of the domain controller
▶ Confirm your input
> The control connects to the Windows domain found.
> The control checks whether all of the required roles have been created in the domain as groups.
▶ Add groups, if necessary
Further information: "Groups of the domain", Page 1857

Joining a Windows domain with a function user

To join a Windows domain with a function user:

▶ Opening the **User administration** window
▶ Select **Connection to Windows domain**
▶ Select the **Active Directory with function user** check box
▶ Select **Find domain**
> The control selects a domain.
▶ Select **Configuration**
▶ Check the data for **Domain name:** and **Key Distribution Center (KDC):**
▶ Enter **Organizational unit for HEROS roles:**
▶ Enter the user name and password of the function user
▶ Press **OK**
▶ Select **APPLY**
> The control connects to the Windows domain found.
> The control checks whether all of the required roles have been created in the domain as groups.

Exporting and importing a Windows configuration file

If you have connected the control to the Windows domain, you can export the required configurations for other controls.

To export the Windows configuration file:

▶ Open the **User administration** window
▶ Select **Connect to Windows domain**
▶ Select **Export the Windows config.**
> The control opens the **Export the Windows domain configuration** window.
▶ Select the directory for the file
▶ Enter the name for the file
▶ Select the **Export the function user's password?** check box, if required
▶ Select **Export**
> The control saves the Windows configuration as a BIN file.

To import the Windows configuration file of another control:

▶ Open the **User administration** window
▶ Select **Connect to Windows domain**
▶ Select **Import the Windows config.**
> The control opens the **Import the Windows domain configuration** window.
▶ Select the existing configuration file
▶ Select the **Import the function user's password?** check box, if required
▶ Select **Import**
> The control adopts the configurations for the Windows domain.

44.5 Autologin **in user administration**

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

- User login

  Further information: "Logging on with user administration", Page 1860

- Configure user administration

  Further information: "Configuring user administration", Page 1846

Requirements

- User administration has been configured
- The user for **Autologin** has been defined

Description of function

With the **Enable autologin** check box in the **User administration** window, you can define a user for autologin.

Further information: "The User administration window", Page 1850

The control then automatically logs this user on and displays the user interface according to the defined rights.

For further authorizations, the control still requires an authentication to be entered.

Further information: "Window for requesting additional rights", Page 1862

44.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

- Automatic user logon

  Further information: "Autologin in user administration", Page 1860

Requirements

- User administration has been configured
- For logon with smartcards:
  - Euchner EKS card reader
  - Smartcard assigned to a user

    Further information: "Assigning a smartcard to a user", Page 1862

Description of function

The control displays the Login dialog in the following cases:

- After the **User logout** function has been executed
- After the **Switch user** function has been executed
- After the screen has been locked by the **screensaver**
- Immediately after control startup if user administration is active and **Autologin** is not enabled

Further information: "HEROS menu", Page 1868

The logon dialog gives you the following options:

- Users who logged in at least once
- **Other** user

Logon with smartcards

You can save a user's logon data on a smartcard and then log the user on with a card reader, without needing to enter a password. You can define whether a PIN is necessary for logon.

The card reader is attached over a USB port. You assign the smartcard to a reader as a token.

Further information: "Assigning a smartcard to a user", Page 1862

The smartcard also has additional memory space, where the machine manufacturer can store his own user-specific data.

44.6.1 Logging on a user with password

To logon a user the first time:

- ▶ Select **Other** in the login dialog
- > The control enlarges the user icon you selected.
- ▶ Enter the user name
- ▶ Enter the user's password

> **ⓘ** The control shows in the Login dialog whether CAPS LOCK is active.

- > The control opens a window with the message **Password expired. Change the password now.**
- ▶ Enter the current password
- ▶ Enter a new password
- ▶ Repeat the new password
- > The control uses the new user to log you in.
- > The control displays this user in the dialog during the next logon procedure.

44.6.2 Assigning a smartcard to a user

To assign a smartcard to a user:
- ▶ Insert a blank smartcard in the card reader
- ▶ Logon the desired smartcard user in user administration
- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **Create token**
- > The control opens the **Write certificate to token** window.
- > The control displays the smartcard in the **Select token** area.
- ▶ Select the smartcard as the token to be written
- ▶ Enable the **PIN protection?** check box, if required
- ▶ Enter user password (and PIN, if desired)
- ▶ Select **Start writing**
- > The control saves the user's logon data on the smartcard.
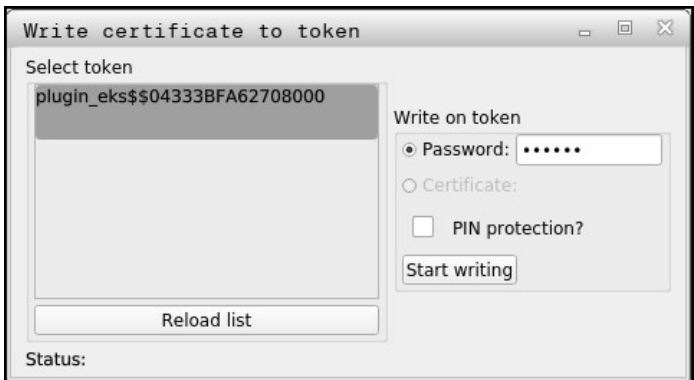
Notes
- You must restart the control in order for it to detect a card reader.
- You can overwrite smartcards that already contain information.
- If you change a user's password, you must reassign the smartcard.

44.7 Window for requesting additional rights

Application

If you do not have the rights required for a specific **HEROS menu** item, the control opens the window for requesting additional rights.

In this window, you can temporarily obtain more rights by adding another user's rights.

Related topics
- Temporarily granting additional rights in the **Active user** window

  Further information: "The Active user window", Page 1850

Description of function

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

You must enter the password in order to enable user rights.
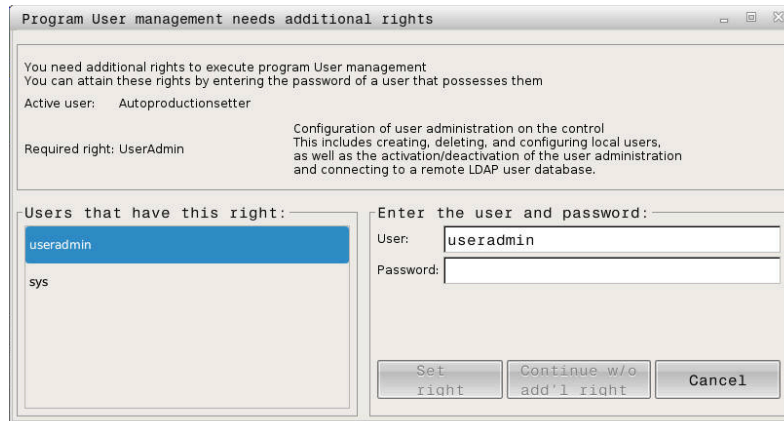
Window for requesting additional rights

To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Notes

- If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
- You can't use this window to change user administration settings. The user with the HEROS.Admin role must be logged on in order to do so.

44.8 SSH-secured DNC connection

Application

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

For DNC connections using the RPC or LSV2 protocol, the connection is routed through an SSH tunnel. This method assigns the remote user to a user set up on the control, granting the remote user this user's rights.

Related topics

- Forbidding non-secure connections
  Further information: "Firewall", Page 1825
- Roles for remote logon
  Further information: "Roles", Page 1844

Requirements

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

Description of function

Concept of transmission through an SSH tunnel

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to protect the connection. This key pair is generated on the client. The key pair consists of a private key and a public key. The private key remains with the client. During setup, the public key is transferred to the server and assigned to a certain user.

The client tries to connect to the server using the pre-defined user name. The server can use the public key to verify that the requester of the connection holds the associated private key. If yes, the server accepts the SSH connection and assigns it to the user that has been used for the login. Communication can then be "tunneled" through this SSH connection.
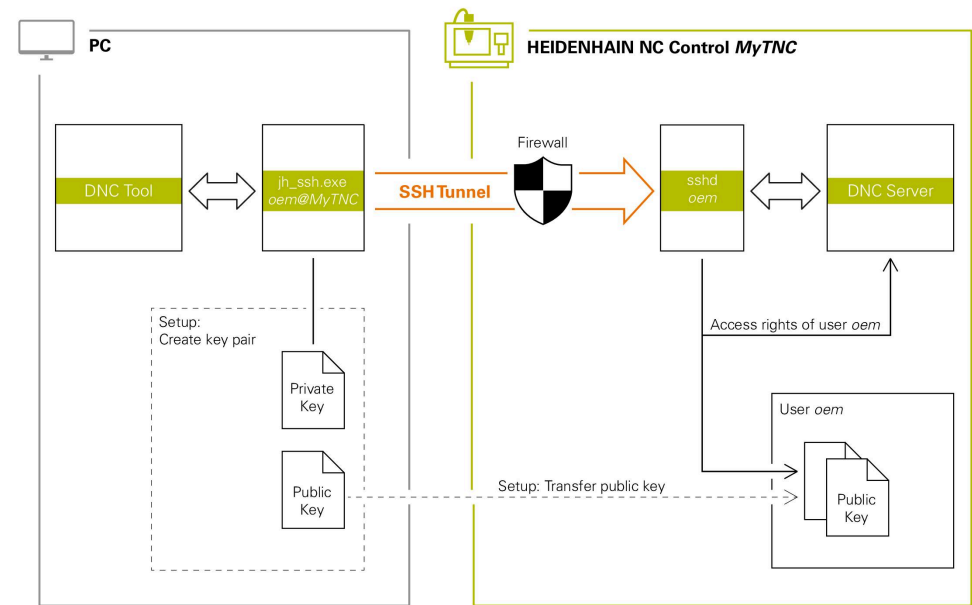
Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, establishing, and managing secure connections through an SSH tunnel.

When the connection is set up, the required key pair is generated in TNCremo and the public key is transferred to the control.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.

> **i** In order to expand the connection configuration using the associated **CreateConnections** tool, you need to update to **HEIDENHAIN DNC v1.7.1**. A modification of the application source code is not required.

44.8.1 Setting up SSH-secured DNC connections

To set up an SSH-secured DNC connection for the logged-on user:

▶ Select the **Settings** application
▶ Select **Network/Remote Access**
▶ Select **DNC**
▶ Activate the **Setup permitted** toggle switch
▶ Use **TNCremo** to set up the secure connection (TCP secure).

> 📖 For details, refer to the integrated help system of TNCremo.

> TNCremo transmits the public key to the control.

> ℹ️ In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

▶ Deactivate the **Setup permitted** toggle switch

44.8.2   Removing a secure connection

If you delete a private key from the control, that user no longer has the possibility of a secure connection.

To delete a key:

▶ Select the **Settings** application

▶ Select **Operating System**

▶ Double-tap or double-click **Current User**

> The control opens the **Active user** window.

▶ Select **Certificate and keys**

▶ Select the key to be deleted

▶ Select **Delete SSH key**

> The control deletes the selected key.

Notes

- The the encryption used with the SSH tunnel protects the communication from attackers.
- For OPC UA connections, a stored user certificate is used for authentication.
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804
- 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 authentication.

  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transfer a public key to the control by using a USB device or network drive.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

45

HEROS Operating System

45.1 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 basic features the version HEROS 5.

45.2 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: "The Settings Application", Page 1777

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 1872
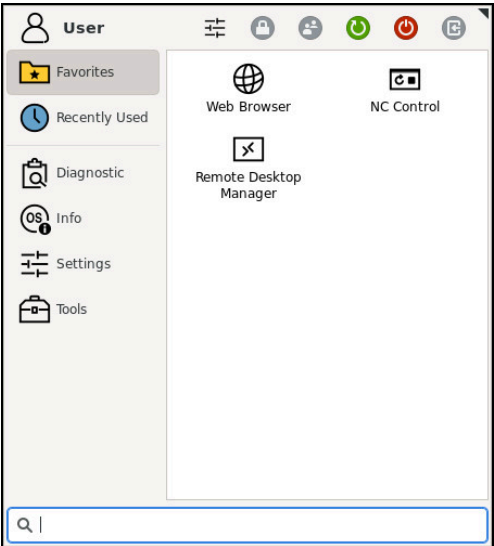
Standard view of the HEROS menu

The HEROS menu contains the following functions:

Area	Function
Header	■ User name
	Further information: "The Active user window", Page 1850
	■ User-specific settings
	■ Lock display
	Only if user administration is active
	■ Switch user
	Only if user administration is active
	■ Restart
	■ Shut down
	■ Log out
	Only if user administration is active
	Further information: "User Administration", Page 1841
Navigation	■ Favorites
	■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists
	■ **HeLogging**: 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 1829
	■ **Portscan OEM**: Available only to authorized specialists
	■ **RemoteService**: Start and stop remote maintenance
	Further information: "Secure Remote Access", Page 1879
	■ **Terminal**: Enter and execute console commands

Area	Function
	■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components (particularly motors) and presents it graphically
	Further information: "TNCdiag", Page 1832
	■ **TNCscope**
	Software for data recording
Settings	■ **Adjust screen brightness**: Adjust screen brightness
	■ **Screensaver**: Screensaver
	■ **Current User**
	Further information: "The Active user window", Page 1850
	■ **Date/Time**
	Further information: "The Adjust system time window", Page 1789
	■ **Firewall**
	Further information: "Firewall", Page 1825
	■ **HePacketManager**: Available only to authorized specialists
	■ **HePacketManager Custom**: Available only to authorized specialists
	■ **Language/Keyboards**
	Further information: "Conversational language of the control", Page 1790
	■ **Network**
	Further information: "Ethernet interface", Page 1795
	■ **OEM Function Users**
	Further information: "User Administration", Page 1841
	■ **OPC UA NC Server Connection Assistant**
	Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 1808
	■ **OPC UA NC Server License**
	Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 1809
	■ **PKI Admin**: Manage certificates for the control, such as for **OPC UA NC Server**
	Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 1804
	■ **Printer**
	Further information: "Printers", Page 1812
	■ **Screenshot Config**
	In the **Screenshot settings** window, you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).
	■ **SELinux**
	Further information: "SELinux security software", Page 1791
	■ **Shares**
	Further information: "Network drives on the control", Page 1792
	■ **UserAdmin**
	Further information: "The User administration window", Page 1850
	■ **VNC**
	Further information: "The VNC menu item", Page 1815
	■ **WindowManagerConfig**: Settings for the Window Manager
	Further information: "Window Manager", Page 1873

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
	■ **Diffuse Merge Tool**: Compare and merge text files

> **ⓘ** To compare NC programs, the control offers the **program comparison** function.
> **Further information:** "Program comparison", Page 1172

	■ **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 1829
	■ **NC/PLC Restore**
	Further information: "Backup and restore", Page 1829
	■ **QupZilla**: Alternative web browser for touch operation
	■ **Real VNC Viewer**: Define the settings for external software accessing the control (e.g., for maintenance purposes)
	■ **Remote Desktop Manager**
	Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 1819
	■ **Ristretto**: Open graphics files
	■ **Secure Remote Access**
	Further information: "Secure Remote Access", Page 1879
	■ **Combine fixtures**
	Further information: "Combining fixtures in the New Fixture window", Page 875
	■ **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 for 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 1869
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 1825
     ■ Network status
       Further information: "Ethernet interface", Page 1795
     ■ 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 **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 91

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 91
- 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.

45.3 Serial data transfer

Application

The TNC7 basic 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 1833

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**
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state.

Machine parameters	Setting
	■ **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.

Further information: "PC software for data transfer", Page 1875

45.4 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 8
   ■ Windows 10
■ PC RAM: 2 GB
■ Free PC hard-disk space: 15 MB
■ A network connection to the control

Description of function

The TNCremo data transfer software provides the following areas:
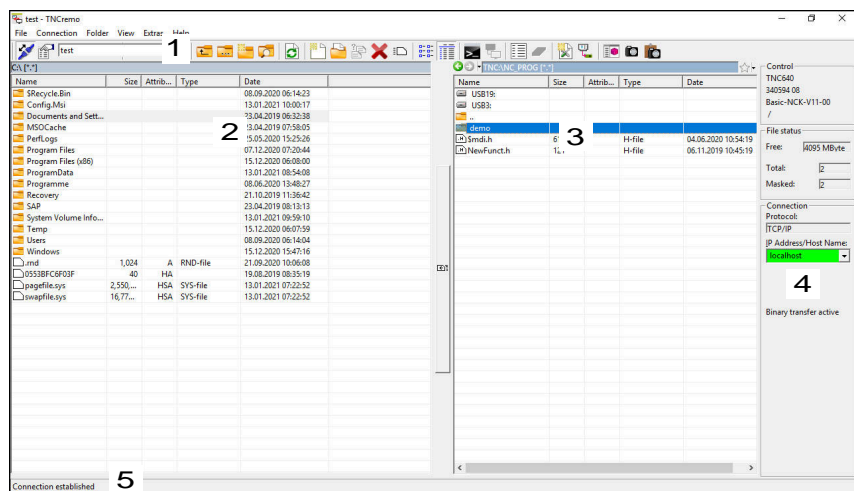
1 Toolbar

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 authentication.

  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

45.5 File transfer with SFTP (SSH File Transfer Protocol)

Application

SFTP (SSH File Transfer Protocol) provides a secure way to connect client applications to the control and to transfer files at high speed from a PC to the control. The connection is routed via an SSH tunnel.

Related topics

- User administration
  Further information: "User Administration", Page 1841
- Principle of the SSH connection
  Further information: "Concept of transmission through an SSH tunnel", Page 1864
- Firewall settings
  Further information: "Firewall", Page 1825

Requirements

- PC software TNCremo with version 3.3 or higher is installed
  Further information: "PC software for data transfer", Page 1875
- **SSH** service is permitted in the firewall of the control
  Further information: "Firewall", Page 1825

Description of function

SFTP is a secure transmission protocol supported by various operating systems for client applications.

To set up the connection, you need a key pair consisting of a public and a private key. You transfer the public key to the control and assign it to a user through the user administration. The private key is required by the client application to set up a connection to the control.

HEIDENHAIN recommends using the CreateConnections application to generate the key pair. CreateConnections is installed together with the PC software TNCremo with version 3.3 and higher. CreateConnections lets you transfer the public key directly to the control and assign it to a user.

You can also use other software to generate the key pair.

45.5.1 Setting up an SFTP connection with CreateConnections

For an SFTP connection using CreateConnections, the following are required:

- Connection with secure protocol, such as **TCP/IP Secure**
- User name and password of the desired user are known

> ℹ️ When you transfer the public key to the control, you must enter the user's password twice.
>
> If user administration is inactive, the user **user** is logged in. The password for the user **user** is **user**.

To set up an SFTP connection:

▶ Select the **Settings** application

▶ Select **Network/Remote Access**

▶ Select **DNC**

▶ Activate the **Setup permitted** toggle switch

▶ Create a key pair with CreateConnections and transfer it to the control

> 📖 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.

▶ Deactivate the **Setup permitted** toggle switch

▶ Transfer the private key to the client application

▶ Connect the client application to the control

> 📖 Please refer to the manual of the client application.

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 authentication. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

- During the connection, the rights of the user to whom the used key is assigned are active. The directories and files displayed, as well as the access options, vary depending on the permissions.

- You can also transfer a public key to the control by using a USB device or network drive. In this case, you do not need to activate the **Allow password authentication** check box.

- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

45.6 Secure Remote Access

Application

Secure Remote Access SRA allows you to set up an encrypted connection between a PC and your control via the Internet. SRA allows the control to be displayed and operated on a PC, such as for service trainings or remote maintenance.

Related topics

- VNC settings

  Further information: "The VNC menu item", Page 1815

Requirements

- Existing Internet connection

  Further information: "Network configuration with Advanced Network Configuration", Page 1883

- The following settings in the **VNC settings** window:

  - **Enable RemoteAccess and IPC** check box is active

  - In the **Enabling other VNC** area, the **Inquire** or **Permitted** check box is active

  Further information: "The VNC menu item", Page 1815

- PC with paid RemoteAccess software including the extension **Secure Remote Access**

  HEIDENHAIN website

> 📖 For more information, refer to the integrated help system of RemoteAccess.
>
> You can open the context-sensitive help function of the RemoteAccess software by pressing the **F1** key.

Description of function

To navigate to this function:

Tools ▶ Secure Remote Access

The PC provides a ten-digit session ID for you to enter in the **HEIDENHAIN Secure Remote Access** window.

SRA enables connection via an VPN server.
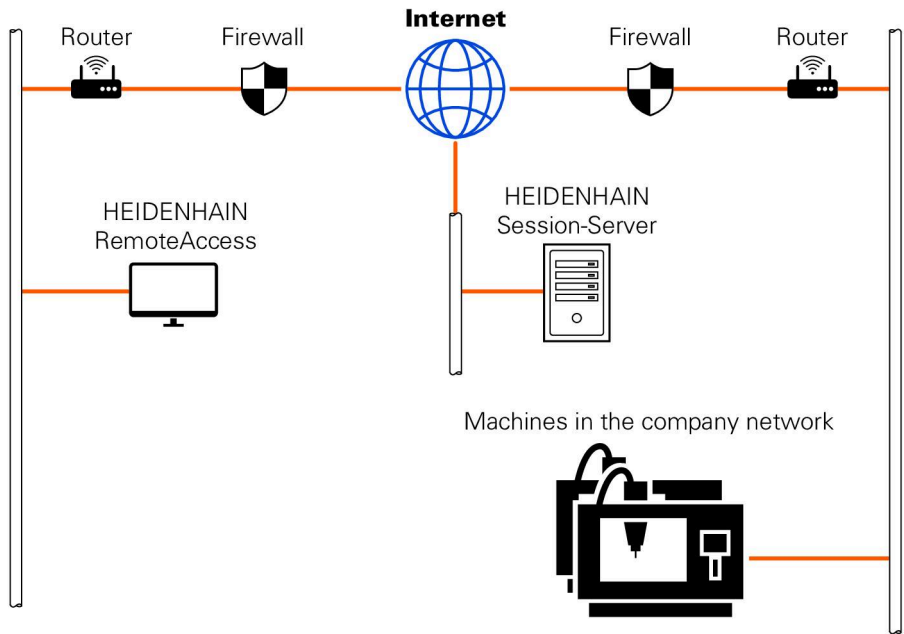
In the **Extended** area, the control shows the progress of the connection setup.

The **HEIDENHAIN Secure Remote Access** window provides the following buttons:

Button	Function
Connect	The control starts the connection with the entered session ID.
Update	The control manually searches for updates for SRA.
	The control automatically searches for available updates when you open the **HEIDENHAIN Secure Remote Access** window.
	If an update is available, you can install it. The control restarts during the update.
Config.	The control opens the **Network settings** window.
	Only for network specialists
Show log	The control opens the log files of the SRA.

Notes

If, in the **VNC settings** window, you set the **Enabling other VNC** setting to **Inquire**, you can permit or deny any connection.

45.7 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 822

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 1829

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 1875
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 1792
- External data carriers
  You can back-up files to external data carriers or use external data carriers to transfer the files.
  Further information: "USB devices", Page 841

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.

45.8 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 829

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS) 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.

45.8.1 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 1791

45.9 Network configuration with Advanced Network Configuration

Application

Use **Advanced Network Configuration** to edit or remove profiles for the network connection.

Related topics

- Network settings

  Further information: "The Editing network connection window", Page 1884

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.
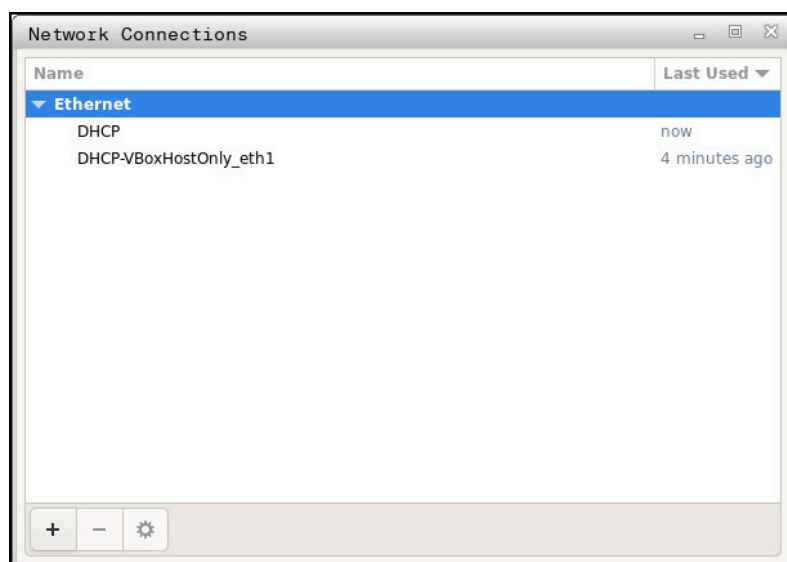
The **Network Connections** window

Symbols in the Network Connections window

The following symbols are shown in the **Network Connections** window:

Icon	Function
✚	Add network connection
▬	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "The Editing network connection window", Page 1884

45.9.1 The 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.
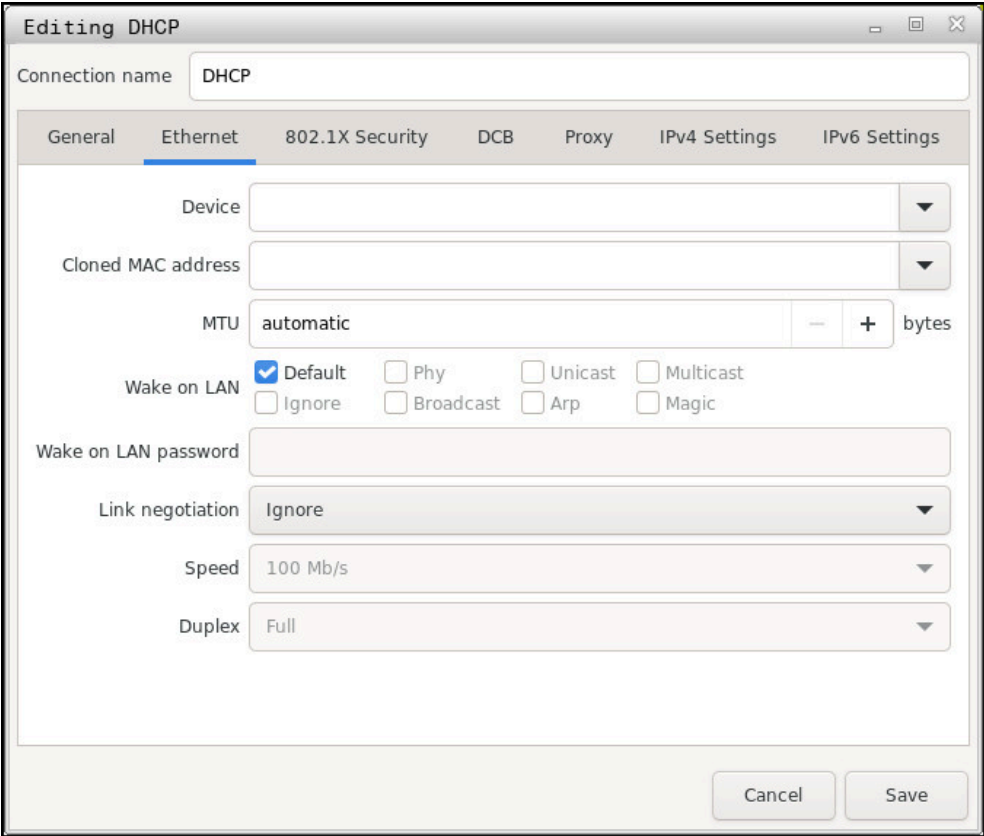
The **Editing network connection** window

The General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically	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 when using this connection	Currently no function
Bonded connections:	Currently no function

The Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Service:	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**
_Private key password:	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 if **Link negotiation** has been selected **Manual**
	Selection by means of a selection window
Full duplex	Here you have to select the duplex setting:
	■ **Half**
	■ **Full**
	Only if **Link negotiation** has been selected **Manual**
	Selection by means of a selection window

The 802.1X Security tab

Currently no function

The DCB tab

Currently no function

The Proxy tab

Currently no function

The 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
Automatic, addresses only	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 **_Method: 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

The IPv6 Settings tab

Currently no function

46

Overviews

46.1 Pin layout and cables for data interfaces

46.1.1 V.24/RS-232-C interface for HEIDENHAIN devices

> **i** 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

46.1.2 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

46.2 Machine parameters

The following list shows the machine parameters that you can edit with the code number 123.

Related topics

- Changing machine parameters with the **MPs for setters** application
  Further information: "Machine parameters", Page 1833

46.2.1 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**		-
📁 **CfgDisplayData** Settings for screen displays	100800	1901
🟨 **axisDisplay** Display sequence and display rules for axes	100810	1901
📁 **x**		-
🟨 **axisKey** Key name of the axis	100810. [Index].01501	1901
🟦 **name** Axis designation	100810. [Index].01502	1902
🟨 **rule** Display rule for the axis	100810. [Index].01503	1902
🟦 **axisDisplayRef** Sequence and rules for display axes before crossing the reference marks	100811	1903
📁 **x**		-
🟨 **axisKey** Key name of the axis	100811. [Index].01501	1903
🟦 **name** Axis designation	100811. [Index].01502	1903
🟨 **rule** Display rule for the axis	100811. [Index].01503	1904
🟨 **positionWinDisplay** Type of position display in the position window	100803	1904
🟨 **statusWinDisplay** Type of position display in the Status workspace	100804	1905
🟦 **axisFeedDisplay** Display of the feed rate in the applications of the Manual operating mode	100806	1906
🟦 **spindleDisplay** Display of spindle position in the position display	100807	1906
🟦 **hidePresetTable** Disable the PRESET MANAGEMENT soft key	100808	1907

Depiction in the configuration editor		MP number	Page
▪ **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	1907
▪ **iconPrioList** Sequence of icons in the display		100813	1907
▪ **compatibilityBits** Settings for display behavior		100815	1908
▪ **axesGridDisplay** Axes as list or group in the position display.		100806	1908
▪ **dashbrdWinDisplay** Type of position display in the status overview of the TNC bar		100817	1909
📁 **CfgPosDisplayPace** Display step for the individual axes		101000	1909
📁 **xx**		-	
▪ **displayPace** Display step for position display in [mm] or [°]		101001	1909
▪ **displayPaceInch** Display step for position display in [inch]		101002	1910
📁 **CfgUnitOfMeasure** Definition of unit of measure in effect for display		101100	1910
▪ **unitOfMeasure** Unit of measure for display and user interface		101101	1911
📁 **CfgProgramMode** Format of the NC programs and cycle display		101200	1911
▪ **programInputMode** MDI: Program entry in HEIDENHAIN Klartext format or ISO format		101201	1911
📁 **CfgDisplayLanguage** Definition of the NC and PLC conversational language		101300	1912
▪ **ncLanguage** NC conversational language		101301	1912
▪ **applyCfgLanguage** Load the language of the NC control		101305	1913
▪ **plcDialogLanguage** PLC conversational language		101302	1913
▪ **plcErrorLanguage** PLC error message language		101303	1914
▪ **helpLanguage** Language for online help		101304	1915

Depiction in the configuration editor		MP number	Page
📁 **CfgStartupData** Behavior during control startup		101500	1915
🟨 **powerInterruptMsg** Acknowledge the Power interrupted message		101501	1916
🟦 **opMode** Operating mode that is switched to when the control has fully booted		101503	1916
🟨 **subOpMode** Submode to be activated for the operating mode entered in 'opMode'		101504	1916
📁 **CfgClockView** Display mode for time of day		120600	1917
🟨 **displayMode** Display mode for time of day on the screen		120601	1917
🟨 **timeFormat** Time format of digital clock		120602	1917
📁 **CfgInfoLine** Link row on/off		120700	1918
🟨 **infoLineEnabled** Enable/disable info line		120701	1918
📁 **CfgGraphics** Settings for 3D simulation graphics		124200	1918
🟨 **modelType** Model type of the 3D simulation graphics		124201	1919
🟨 **modelQuality** Model quality of the 3D simulation graphics		124202	1919
🟦 **clearPathAtBlk** Reset tool paths for new BLK FORM		124203	1920
🟦 **extendedDiagnosis** Write graphics journal files after restart		124204	1920
📁 **CfgPositionDisplay** Settings for the digital readout		124500	1920
🟦 **progToolCallDL** Position display with TOOL CALL DL		124501	1921
📁 **CfgTableEditor** Table editor configuration		125300	1921
🟨 **deleteLoadedTool** Behavior when deleting tools from the pocket table		125301	1921
🟨 **indexToolDelete** Behavior when deleting a tool's index entries		125302	1922
📁 **CfgDisplayCoordSys** Setting the coordinate systems for the display		127500	1922

Depiction in the configuration editor			MP number	Page
		transDatumCoordSys	127501	1922
		Coordinate system for the datum shift		
	CfgRemoteDesktop		100800	1923
	Settings for Remote Desktop connections			
		connections	133501	1923
		List of Remote Desktop connections to be displayed		
		autoConnect	133505	1923
		Start connection automatically		
		title	133502	1924
		Name of the OEM operating mode		
		dialogRes	00501	1924
		Name of a text		
		text	00502	1924
		Language-sensitive text		
		icon	133503	1925
		Path/name for optional icon graphic file		
		locations	133504	1925
		List with positions where this Remote Desktop connection is displayed		
		x	-	
		opMode	133504. [Index].133401	1925
		Operating mode		
		subOpMode	133504. [Index].133402	1926
		Optional submode for the operating mode specified in 'opMode'		
	PalletSettings		-	
	CfgPalletBehaviour		202100	1927
	Behavior of the pallet control cycle			
		failedCheckReact	202106	1927
		Specify reaction to program check and tool check		
		failedCheckImpact	202107	1927
		Specify effect of program check or tool check		
	ProbeSettings		-	
	CfgTT		122700	1928
	Configuration of the tool calibration			
		TT140_x	-	
		spindleOrientMode	122704	1928
		M function for spindle orientation		
		probingRoutine	122705	1928
		Probing routine		
		probingDirRadial	122706	1929

Depiction in the configuration editor	MP number	Page
☐ Probing direction for tool-radius measurement		
☐ **offsetToolAxis** Distance from lower edge of tool to upper edge of stylus	122707	1929
☐ **rapidFeed** Rapid traverse in probing cycle for TT tool touch probe	122708	1929
☐ **probingFeed** Probing feed rate for tool measurement with non-rotating tool	122709	1930
☐ **probingFeedCalc** Calculation of the probing feed rate	122710	1930
☐ **spindleSpeedCalc** Speed determination method	122711	1930
☐ **maxPeriphSpeedMeas** Maximum permissible surface speed of the cutting edge for radius measurement	122712	1931
☐ **maxSpeed** Maximum permissible speed during tool measurement	122714	1931
☐ **measureTolerance1** Maximum permissible measuring error for tool measurement with rotating tool (first measuring error)	122715	1931
☐ **measureTolerance2** Maximum permissible measuring error for tool measurement with rotating tool (second measuring error)	122716	1932
☐ **stopOnCheck** NC Stop during "tool check"	122717	1932
☐ **stopOnMeasurement** NC stop during tool measurement	122718	1932
☐ **adaptToolTable** Change the tool table during tool check and tool measurement	122719	1933
📁 **CfgTTRoundStylus** Configuration of a round stylus	114200	1933
📁 **TT140_x**		-
☐ **centerPos** Coordinates of the probe-contact center point	114201	1933
☐ **safetyDistToolAx**	114203	1934

Depiction in the configuration editor	MP number	Page
⬜ Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction		
⬜ **safetyDistStylus** Safety zone around the stylus for pre-positioning	114204	1934
📁 **CfgTTRectStylus** Configuration of a rectangular stylus	114300	1935
📁 **TT140_x**		-
⬜ **centerPos** Coordinates of the stylus center	114313	1935
⬜ **safetyDistToolAx** Set-up clearance above the stylus for pre-positioning	114317	1935
⬜ **safetyDistStylus** Safety zone around the stylus for pre-positioning	114318	1935
📁 **ChannelSettings**		-
📁 **CH_xx**		-
📁 **CfgActivateKinem** Active kinematics	204000	1937
⬜ **kinemToActivate** Kinematics to be activated / active kinematics	204001	1937
🟦 **kinemAtStartup** The kinematics to be activated during control start-up	204002	1937
📁 **CfgNcPgmBehaviour** Specify the behavior of the NC program.	200800	1938
⬜ **operatingTimeReset** Reset the machining time when program starts.	200801	1938
🟦 **plcSignalCycle** PLC signal for the number of the pending machining cycle	200803	1938
🟦 **plcSignalCycState** PLC signal for type of current cycle execution	200805	1938
📁 **CfgGeoTolerance** Geometry tolerances	200900	1939
⬜ **circleDeviation** Permissible deviation of the radius	200901	1939
🟦 **threadTolerance**	200902	1939

Depiction in the configuration editor	MP number	Page
📁 **CfgEditorSettings** Settings for the NC editor	105400	1947
🟨 **createBackup** Generate a backup file *.bak	105401	1947
🟨 **deleteBack** Behavior of the cursor after deletion of lines	105402	1947
🟨 **lineBreak** Line break on NC blocks with more than one line	105404	1948
🟦 **stdTNChelp** Activate help graphics when entering cycle data	105405	1948
🟦 **warningAtDEL** Confirmation request when deleting an NC block.	105407	1948
🟦 **maxLineGeoSearch** Line number up to which a test of the NC program is to be run.	105408	1949
🟦 **blockIncrement** ISO programming: Block number increment	105409	1949
🟦 **useProgAxes** Specify programmable axes	105410	1949
🟦 **enableStraightCut** Allow or lock paraxial positioning blocks	105411	1950
🟦 **noParaxMode** Hide FUNCTION PARAXCOMP/PARAXMODE	105413	1950
🟦 **quotePaths** Put all path information in quotation marks	105414	1951
📁 **CfgPgmMgt** Settings for the file management	122100	1951
🟨 **dependentFiles** Display of dependent files	122101	-
📁 **CfgProgramCheck** Settings for tool-usage files	129800	1952
🟦 **autoCheckTimeOut** Timeout for creation of tool-usage files	129803	1952
🟦 **autoCheckPrg** Create tool-usage file for NC program	129801	1952
🟦 **autoCheckPal** Create pallet-usage files	129802	1953
📁 **CfgUserPath** Paths for the end user	102200	1954
🟨 **ncDir** List of drives and/or directories	102201	1954
🟦 **fn16DefaultPath**	102202	1954

Depiction in the configuration editor	MP number	Page
📁 User settings for component monitoring		
🟨 **enforceReaction**	129401	1962
The configured error reactions are enforced		
🟨 **showWarning**	129402	1962
Display warnings of monitoring tasks		
📁 **CfgMachineInfo**	131700	1963
General information of the machine operator		
🟨 **machineNickname**	131701	1963
Custom name (nickname) of the machine		
🟨 **inventoryNumber**	131702	1963
Inventory number or ID		
🟨 **image**	131703	1963
Photo or image of the machine		
🟨 **location**	131704	1964
Machine location		
🟨 **department**	131705	1964
Department or division		
🟨 **responsibility**	131706	1964
Responsible for the machine		
🟨 **contactEmail**	131707	1965
Contact email address		
🟨 **contactPhoneNumber**	131708	1965
Contact phone number		

46.2.2 Details about the user parameters

> **ℹ** 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 availability information indicates in which software version the machine parameter was introduced.
> - Data objects are not intended for configuration; instead, they structure or group the machine parameters.
>   **Further information:** "Icons and buttons", Page 1837
> - The entry after iTNC shows the machine parameter number on the iTNC 530.

DisplaySettings

CfgDisplayData   100800

Settings for screen displays

Path:	System ► DisplaySettings ► CfgDisplayData		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axisDisplay   100810

Display sequence and display rules for axes

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplay		
Available as of:	597110-08		
Format:	List		
Input:	List (vacant or index 0 to 23) Specifies the sequence and the rules for the display of axes. The top-most entry corresponds to the top-most position. Up to 24 entries with the parameters - axisKey - name - rule		
Default:	–	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axisKey   100810. [Index].01501

Key name of the axis

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplay ► [Index] ► axisKey		
Available as of:	597110-08		
Format:	String		
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 object **CfgAxis** and displayed as a selection menu.		
Default:	–	Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

name			100810.[Index].01502

Axis designation

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplay ► [Index] ► name		
Available as of:	597110-08		
Format:	String		
Input:	max. 2 Characters		
	Define the axis designation that, as an alternative to the key name from **CfgAxis** is used for the display. If the parameter is not set, then the TNC7 basic displays the key name.		
Default:	No value	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

rule			100810.[Index].01503

Display rule for the axis

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplay ► [Index] ► rule		
Available as of:	597110-08		
Format:	Selection		
Input:	Defines 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		
	Axis is shown only if it is used as an axis or a 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.		
Default:	Always	Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axisDisplayRef			100811

Sequence and rules for display axes before crossing the reference marks

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef		
Available as of:	597110-08		
Format:	List		
Input:	List (vacant or index 0 to 23)		
	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, then the entries from the **axisDisplay** (100810) parameter are used. The top-most entry corresponds to the top-most position.		
	Up to 24 entries with the parameters		
	■ axisKey		
	■ name		
	■ rule		
Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axisKey			100811. [Index].01501

Key name of the axis

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef ► [Index] ► axisKey		
Available as of:	597110-08		
Format:	String		
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 object **CfgAxis** and displayed as a selection menu.		
Default:	–	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

name			100811. [Index].01502

Axis designation

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef ► [Index] ► name		

Available as of:	597110-08
Format:	String
Input:	max. 2 Characters
	Define the axis designation that, as an alternative to the key name from **CfgAxis** is used for the display. If the parameter is not set, then the TNC7 basic displays the key name.

Default:	No value	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

rule 100811.
[Index].01503

Display rule for the axis

Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef ► [Index] ► rule
Available as of:	597110-08
Format:	Selection
Input:	Specifies the condition for displaying the axis.
	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
	Axis is shown only if it is used as an axis or a 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.

Default:	Always	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

positionWinDisplay 100803

Type of position display in the position window

Path:	System ► DisplaySettings ► CfgDisplayData ► positionWinDisplay
Available as of:	597110-01
Format:	Selection

Input:	Position display in the position window (positions display 1):
	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 super-impositioning (M118)

Default:	ACTL	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

statusWinDisplay	100804

Type of position display in the Status workspace

Path:	System ► DisplaySettings ► CfgDisplayData ► statusWinDisplay
Available as of:	597110-0
Format:	Selection
Input:	Position display in the status window (position display 2):
	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 super-impositioning (M118)

Default:	ACTL	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axisFeedDisplay 100806

Display of the feed rate in the applications of the **Manual** operating mode

Path:	System ► DisplaySettings ► CfgDisplayData ► axisFeedDisplay		
Available as of:	597110-02		
Format:	Selection		
Input:	**at axis key**		
	Display of the feed rate only if an axis direction key is pressed. The axis-specific feed rate from the machine parameter CfgFeedLimits/**manualFeed** (400304) is shown.		
	always minimum		
	Display of the feed rate for all axes, including before an axis direction key is pressed (lowest value from CfgFeedLimits/**MP_manualFeed**).		
Default:	at axis key	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7270		

spindleDisplay 100807

Display of spindle position in the position display

Path:	System ► DisplaySettings ► CfgDisplayData ► spindleDisplay		
Available as of:	597110-02		
Format:	Selection		
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 an M5 is pending		
	during closed loop or M5 or tapping		
	Display of spindle position if the spindle is servo-controlled or if an M5 is pending, or during a tapping operation		
Default:	during closed loop	Optional:	✓

hidePresetTable 100808

Disable the **PRESET MANAGEMENT** soft key

Path:	System ► DisplaySettings ► CfgDisplayData ► hidePresetTable
Available as of:	597110-03
Format:	Selection menu
Input:	**TRUE** Access to the preset table is locked; the soft key is dimmed **FALSE** The preset table can be accessed via soft key
Default:	FALSE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

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
Available as of:	
Format:	Selection
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.
Default:	FONT_APPLI-CATION_MEDIUM	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

iconPrioList 100813

Sequence of icons in the display

Path:	System ► DisplaySettings ► CfgDisplayData ► iconPrioList
Available as of:	597110-15
Format:	Selection
Input:	**BASIC_ROT** **ROT_3D**

TCPM

ACC

TURNING

AFC

S_PULSE

MIRROR

GPS

RADCORR

PARAXCOMP

MON_FS_OVR

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

compatibilityBits   100815

Settings for display behavior

Path:	System ► DisplaySettings ► CfgDisplayData ► compatibilityBits
Available as of:	597110-14
Format:	Binary value
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.

Default:	%0	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

axesGridDisplay   100816

Axes as list or group in the position display.

Path:	System ► DisplaySettings ► CfgDisplayData ► axesGridDisplay
Available as of:	597110-16
Format:	Numerical value
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

Default:	0	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7270		

dashbrdWinDisplay   100817

Type of position display in the status overview of the TNC bar

Path:	System ► DisplaySettings ► CfgDisplayData ► dashbrdWinDisplay		
Available as of:	597110-18		
Format:	Selection		
Input:	**NOML**		
	ACTL		
	REF ACTL		
	REF NOML		
	LAG		
	ACTDST		
	REFDST		
	M118		
Default:	ACTL	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgPosDisplayPace   101000

Display step for the individual axes

Path:	System ► DisplaySettings ► CfgPosDisplayPace		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

displayPace   101001

Display step for position display in [mm] or [°]

Path:	System ► DisplaySettings ► CfgPosDisplayPace ► [Key name of the axis] ► displayPace		
Available as of:	597110-01		

Format:	Selection		
Input:	**0.1**		
	0.05		
	0.01		
	0.005		
	0.001		
	0.0005		
	0.0001		
	0.00005		
	0.00001		
	0.000005		
	0.000001		
Default:	0.001	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7290.0-8		

displayPaceInch   101002

Display step for position display in [inch]

Path:	System ► DisplaySettings ► CfgPosDisplayPace ► [Key name of the axis] ► displayPaceInch		
Available as of:	597110-01		
Format:	Selection		
Input:	**0.005**		
	0.001		
	0.0005		
	0.0001		
	0.00005		
	0.00001		
	0.000005		
	0.000001		
Default:	0.001	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7290.0-8		

CfgUnitOfMeasure   101100

Definition of unit of measure in effect for display

Path:	System ► DisplaySettings ► CfgUnitOfMeasure

Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

unitOfMeasure   101101

Unit of measure for display and user interface

Path:	System ► DisplaySettings ► CfgUnitOfMeasure ► unitOfMeasure		
Available as of:	597110-01		
Format:	Selection		
Input:	**metric**		
	Metric measurement system		
	inch		
	Inches		
Default:	metric	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgProgramMode   101200

Format of the NC programs and cycle display

Path:	System ► DisplaySettings ► CfgProgramMode		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

programInputMode   101201

MDI: Program entry in HEIDENHAIN Klartext format or ISO format

Path:	System ► DisplaySettings ► CfgProgramMode ► programInputMode		
Available as of:	597110-01		
Format:	Selection		
Input:	**HEIDENHAIN**		
	Program entry with HEIDENHAIN Klartext		
	ISO		
	Program entry according to ISO		
Default:	HEIDENHAIN	Optional:	-

| Reaction: | Program run is locked (RUN) | Access: | LEVEL1 |

CfgDisplayLanguage 101300

Definition of the NC and PLC conversational language

Path:	System ► DisplaySettings ► CfgDisplayLanguage		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

ncLanguage 101301

NC conversational language

Path:	System ► DisplaySettings ► CfgDisplayLanguage ► ncLanguage	
Available as of:	597110-0	
Format:	Selection	
Input:	**ENGLISH**	
	GERMAN	
	CZECH	
	FRENCH	
	ITALIAN	
	SPANISH	
	PORTUGUESE	
	SWEDISH	
	DANISH	
	FINNISH	
	DUTCH	
	POLISH	
	HUNGARIAN	
	RUSSIAN	
	CHINESE	
	CHINESE_TRAD	
	SLOVENIAN	
	KOREAN	
	NORWEGIAN	
	ROMANIAN	
	SLOVAK	
	TURKISH	

Default:	ENGLISH	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7230.0		

applyCfgLanguage   101305

Load the language of the NC control

Path:	System ► DisplaySettings ► CfgDisplayLanguage ► applyCfgLanguage
Available as of:	597110-14
Format:	Selection menu
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.

Default:	FALSE	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

plcDialogLanguage   101302

PLC conversational language

Path:	System ► DisplaySettings ► CfgDisplayLanguage ► plcDialogLanguage
Available as of:	597110-0
Format:	Selection
Input:	**ENGLISH**
	GERMAN
	CZECH
	FRENCH
	ITALIAN
	SPANISH
	PORTUGUESE
	SWEDISH
	DANISH
	FINNISH
	DUTCH
	POLISH
	HUNGARIAN
	RUSSIAN
	CHINESE
	CHINESE_TRAD

SLOVENIAN

KOREAN

NORWEGIAN

ROMANIAN

SLOVAK

TURKISH

Default:	ENGLISH	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7230.1		

plcErrorLanguage		101303
PLC error message language		

Path:	System ► DisplaySettings ► CfgDisplayLanguage ► plcErrorLanguage
Available as of:	597110-0
Format:	Selection
Input:	**ENGLISH**
	GERMAN
	CZECH
	FRENCH
	ITALIAN
	SPANISH
	PORTUGUESE
	SWEDISH
	DANISH
	FINNISH
	DUTCH
	POLISH
	HUNGARIAN
	RUSSIAN
	CHINESE
	CHINESE_TRAD
	SLOVENIAN
	KOREAN
	NORWEGIAN
	ROMANIAN
	SLOVAK
	TURKISH

Default:	ENGLISH	Optional:	-

Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7230.2		

helpLanguage 101304

Language for online help

Path:	System ► DisplaySettings ► CfgDisplayLanguage ► helpLanguage		
Available as of:	597110-0		
Format:	Selection		
Input:	**ENGLISH**		
	GERMAN		
	CZECH		
	FRENCH		
	ITALIAN		
	SPANISH		
	PORTUGUESE		
	SWEDISH		
	DANISH		
	FINNISH		
	DUTCH		
	POLISH		
	HUNGARIAN		
	RUSSIAN		
	CHINESE		
	CHINESE_TRAD		
	SLOVENIAN		
	KOREAN		
	NORWEGIAN		
	ROMANIAN		
	SLOVAK		
	TURKISH		
Default:	ENGLISH	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7230.3		

CfgStartupData 101500

Behavior during control startup

Path:	System ► DisplaySettings ► CfgStartupData		

Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

powerInterruptMsg   101501

Acknowledge the **Power interrupted** message

Path:	System ► DisplaySettings ► CfgStartupData ► powerInterruptMsg		
Available as of:	597110-01		
Format:	Selection menu		
Input:	**TRUE** Start-up is only continued after the message has been acknowledged. **FALSE** The **Power interrupted** message does not appear		
Default:	FALSE	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

opMode   101503

Operating mode that is switched to when the control has fully booted

Path:	System ► DisplaySettings ► CfgStartupData ► opMode		
Available as of:	597110-16		
Format:	String		
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		
Default:		Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

subOpMode   101504

Submode to be activated for the operating mode entered in 'opMode'

Path:	System ► DisplaySettings ► CfgStartupData ► subOpMode		
Available as of:	597110-16		
Format:	String		

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

Default:		Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgClockView 120600

Display mode for time of day

Path:	System ► DisplaySettings ► CfgClockView
Available as of:	597110-0
Format:	List
Data object:	

Default:		Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

displayMode 120601

Display mode for time of day on the screen

Path:	System ► DisplaySettings ► CfgClockView ► displayMode
Available as of:	597110-05
Format:	Selection

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

Default:	Analog	Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

timeFormat 120602

Time format of digital clock

Path:	System ► DisplaySettings ► CfgClockView ► timeFormat		
Available as of:	597110-16		
Format:	Selection		
Input:	Possible settings:		
	12 h format		
	Time in 12 hours format		
	24 h format		
	Time in 24 hours format		
Default:	24 h format	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgInfoLine   120700

Link row on/off

Path:	System ► DisplaySettings ► CfgInfoLine		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

infoLineEnabled   120701

Enable/disable info line

Path:	System ► DisplaySettings ► CfgInfoLine ► infoLineEnabled		
Available as of:	597110-05		
Format:	Selection		
Input:	**OFF**		
	The info line is disabled		
	ON		
	The info line below the operating mode display is enabled		
Default:	ON	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgGraphics   124200

Settings for 3D simulation graphics

Path:	System ► DisplaySettings ► CfgGraphics		
Available as of:	597110-0		

Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

modelType 124201

Model type of the 3D simulation graphics

Path:	System ► DisplaySettings ► CfgGraphics ► modelType		
Available as of:	597110-08		
Format:	Selection		
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)		
Default:	3D	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

modelQuality 124202

Model quality of the 3D simulation graphics

Path:	System ► DisplaySettings ► CfgGraphics ► modelQuality		
Available as of:	597110-08		
Format:	Selection		
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		
Default:	medium	Optional:	-

| Reaction: | Change at any time (NOTHING) | Access: | LEVEL1 |

clearPathAtBlk   124203

Reset tool paths for new BLK FORM

Path:	System ► DisplaySettings ► CfgGraphics ► clearPathAtBlk		
Available as of:	597110-12		
Format:	Selection		
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		
Default:		Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

extendedDiagnosis   124204

Write graphics journal files after restart

Path:	System ► DisplaySettings ► CfgGraphics ► modelType		
Available as of:	597110-14		
Format:	Selection menu		
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.		
Default:	OFF	Optional:	✓
Reaction:	Shut down the control (RESET)	Access:	LEVEL1

CfgPositionDisplay   124500

Settings for the digital readout

Path:	System ► DisplaySettings ► CfgPositionDisplay		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-

| Reaction: | Program run is locked (RUN) | Access: | LEVEL1 |

progToolCallDL 124501

Position display with TOOL CALL DL

Path:	System ► DisplaySettings ► CfgPositionDisplay ► progToolCallDL		
Available as of:	597110-09		
Format:	Selection		
Input:	**As Tool Length**		
	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		
	The programmed oversize DL in the TOOL CALL block is not taken into account in the nominal position display. It therefore has the effect of a workpiece oversize.		
Default:	As Tool Length	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgTableEditor 125300

Table editor configuration

Path:	System ► TableSettings ► CfgTableEditor		
Available as of:	597110-12		
Format:	List		
Data object:	Specifies properties and settings for the table editor.		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

deleteLoadedTool 125301

Behavior when deleting tools from the pocket table

Path:	System ► TableSettings ► CfgTableEditor ► deleteLoadedTool	
Available as of:	597110-12	
Format:	Selection	
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

Default:	DISABLED	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	7263 Bit4, 7263 Bit5		

indexToolDelete   125302

Behavior when deleting a tool's index entries

Path:	System ► TableSettings ► CfgTableEditor ► indexToolDelete		
Available as of:	597110-12		
Format:	Selection		
Input:	Possible settings: **ALWAYS_ALLOWED** Deletion of index entries is always possible **TOOL_RULES** Behavior depends on the setting of the parameter delete-LoadedTool		
Default:	ALWAYS_ALLOWED	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	7263 Bit6		

CfgDisplayCoordSys   127500

Setting the coordinate systems for the display

Path:	System ► DisplaySettings ► CfgDisplayCoordSys		
Available as of:	597110-12		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

transDatumCoordSys   127501

Coordinate system for the datum shift

Path:	System ► DisplaySettings ► CfgDisplayCoordSys ► transDatumCoordSys		
Available as of:	597110-12		
Format:	Selection		
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)

Default:	No value	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgRemoteDesktop 133500

Settings for Remote Desktop connections

Path:	System ► DisplaySettings ► CfgRemoteDesktop		
Available as of:	597110-16		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

connections 133501

List of Remote Desktop connections to be displayed

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► connections		
Available as of:	597110-16		
Format:	String		
Input:	Enter here the name of a RemoteFX connection from Remote Desktop Manager. max. 80 Characters		
Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

autoConnect 133505

Start connection automatically

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► autoConnect		
Available as of:	597110-17		
Format:	Selection menu		
Input:	**TRUE** Automatically connect when control boots **FALSE** Do not start connection automatically.		

Default:	TRUE	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

title 133502

Name of the OEM operating mode

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► title
Available as of:	597110-16
Format:	List
Input:	Specifies the name of the OEM operating mode for display on the TNC and in the information bar.
Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

dialogRes 00501

Name of a text

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► title ► dialogRes
Available as of:	597110-16
Format:	String
Input:	max. 40 Characters
	The text must be available with this name in a text resource file.
	If the text is not intended to be language-sensitive, leave machine parameter **dialogRes** (00501) empty. Then enter the text in the machine parameter **text** (00502).
	Starting with software -17: If the text comes from a *.po file, the machine parameter **poDomain** (00504) must also be filled in.
Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

text 00502

Language-sensitive text

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► title ► text
Available as of:	597110-16
Format:	String
Input:	max. 60 Characters

This text is loaded from a text resource file and should not be changed here.

If the text is not language-specific, enter it here directly. In this case, do not enter anything in the machine parameter **dialogRes** (606202).

Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

icon

133503

Path/name for optional icon graphic file

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► icon		
Available as of:	597110-16		
Format:	String		
Input:	max. 260 Characters		
Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

locations

133504

List with positions where this Remote Desktop connection is displayed

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► locations		
Available as of:	597110-16		
Format:	List		
Input:			
Default:	–	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

opMode

133504. [Index].133401

Operating mode

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► locations ► [Index] ► opMode		
Available as of:	597110-16		
Format:	String		
Input:	max. 80 Characters		
Default:	–	Optional:	-

| Reaction: | Restart the control (REBOOT) | Access: | LEVEL1 |

subOpMode   133504.
                                                                        [Index].133402

Optional submode for the operating mode specified in 'opMode'

Path:	System ► DisplaySettings ► CfgRemoteDesktop ► locations ► [Index] ► subOpMode		
Available as of:	597110-16		
Format:	String		
Input:	max. 80 Characters		
Default:	–	Optional:	-
Reaction:	Restart the control (REBOOT)	Access:	LEVEL1

PalletSettings

CfgPalletBehaviour		202100
Behavior of the pallet control cycle		
Path:	System ► PalletSettings ► CfgPalletBehaviour	
Available as of:	597110-0	
Format:	List	
Data object:		
Default:		Optional: -
Reaction:	Program run is locked (RUN)	Access: LEVEL1

failedCheckReact		202106
Specify reaction to program check and tool check		
Path:	System ► PalletSettings ► CfgPalletBehaviour ► failedCheckReact	
Available as of:	597110-17	
Format:	Selection	
Input:	**Never**	
	No checking for faulty program or tool calls.	
	OnFailedPgmCheck	
	Check for faulty program calls.	
	OnFailedToolCheck	
	Check for faulty tool calls.	
Default:	Never	Optional: ✓
Reaction:	Program run is locked (RUN)	Access: LEVEL1

failedCheckImpact		202107
Specify effect of program check or tool check		
Path:	System ► PalletSettings ► CfgPalletBehaviour ► failedCheckImpact	
Available as of:	597110-17	
Format:	Selection	
Input:	**SkipPGM**	
	Skip faulty programs.	
	SkipFIX	
	Skip fixture setups that contain faulty programs.	
	SkipPAL	
	Skip pallets that contain faulty programs.	
Default:	SkipPGM	Optional: ✓
Reaction:	Program run is locked (RUN)	Access: LEVEL1

ProbeSettings

CfgTT		122700
Configuration of the tool calibration		

Path:	System ► ProbeSettings ► CfgTT	
Available as of:	597110-0	
Format:	List	
Data object:		
Default:		Optional: -
Reaction:	Change at any time (NOTHING)	Access: LEVEL1

spindleOrientMode		122704
M function for spindle orientation		

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► spindleOrientMode	
Available as of:	597110-03	
Format:	Numerical value	
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

Default:	-1	Optional: -
Reaction:	Change at any time (NOTHING)	Access: LEVEL1
iTNC 530:	MP6560	

probingRoutine		122705
Probing routine		

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► probingRoutine	
Available as of:	597110-05	
Format:	Selection	
Input:	**MultiDirections** The probe contact is probed from several directions. **SingleDirection** The probe contact is probed from one direction.	
Default:	MultiDirections	Optional: -

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6500 Bit 8		

probingDirRadial			122706

Probing direction for tool-radius measurement

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► probingDirRadial		
Available as of:	597110-03		
Format:	Selection		
Input:	**X_Positive**		
	Y_Positive		
	X_Negative		
	Y_Negative		
	Z_Positive		
	Z_Negative		
Default:	X_Positive	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-03		
Format:	Numerical value		
Input:	0.001 to 99.9999, max. 4 decimal places		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-03		
Format:	Numerical value		
Input:	10 to 300000		

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-03		
Format:	Numerical value		
Input:	1 to 3000		
Default:	60 mm/min or inches/min	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6520		

probingFeedCalc   122710

Calculation of the probing feed rate

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► probingFeedCalc		
Available as of:	597110-03		
Format:	Selection		
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		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6507		

spindleSpeedCalc   122711

Speed determination method

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► spindleSpeedCalc		
Available as of:	597110-03		
Format:	Selection		
Input:	**Automatic**		

Automatically determine speed

MinSpindleSpeed

Always use minimum spindle speed

Default:	Automatic	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6500 Bit4		

maxPeriphSpeedMeas 122712

Maximum permissible surface speed of the cutting edge for radius measurement

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► maxPeriphSpeedMeas		
Available as of:	597110-03		
Format:	Numerical value		
Input:	1 to 129, max. 4 decimal places		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6570		

maxSpeed 122714

Maximum permissible speed during tool measurement

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► maxSpeed		
Available as of:	597110-03		
Format:	Numerical value		
Input:	0 to 1000		
Default:	0 rpm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6572		

measureTolerance1 122715

Maximum permissible measuring error for tool measurement with rotating tool (first measuring error)

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► measureTolerance1		
Available as of:	597110-03		
Format:	Numerical value		
Input:	0.001 to 0.999, max. 3 decimal places		
Default:		Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6510.0		

measureTolerance2 122716

Maximum permissible measuring error for tool measurement with rotating tool (second measuring error)

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► measureTolerance2		
Available as of:	597110-0		
Format:	Numerical value		
Input:	0.001 to 0.999, max. 3 decimal places		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6510.1		

stopOnCheck 122717

NC Stop during "tool check"

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► stopOnCheck		
Available as of:	597110-03		
Format:	Selection menu		
Input:	**TRUE** If the breakage tolerance is exceeded, the NC program stops and the error message **Tool broken** is displayed. **FALSE** The NC program does not stop if the breakage tolerance is exceeded.		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6500 Bit5		

stopOnMeasurement 122718

NC stop during tool measurement

Path:	System ► ProbeSettings ► CfgTT ► [Key name of the TT] ► stopOnMeasurement		
Available as of:	597110-03		
Format:	Selection menu		
Input:	**TRUE**		

If the breakage tolerance is exceeded, the NC program stops and the error message **Touch point inaccessible** is displayed.

FALSE

The NC program does not stop if the breakage tolerance is exceeded.

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-03		
Format:	Selection		
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.		
Default:	AdaptOnBoth	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6500 Bit11		

CfgTTRoundStylus   114200

Configuration of a round stylus

Path:	System ► ProbeSettings ► CfgTTRoundStylus		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

centerPos   114201

Coordinates of the probe-contact center point

Path:	System ► ProbeSettings ► CfgTTRoundStylus ► [Key name of the TT] ► centerPos
Available as of:	597110-03
Format:	Numerical value
Input:	-99999.9999 to 99999.9999 [mm], max. 4 decimal places
	Coordinates of the probe-contact center with respect to the machine datum.
	■ [0]: X coordinate
	■ [1]: Y coordinate
	■ [2]: Z coordinate

Default:	0 mm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

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
Available as of:	597110-03
Format:	Numerical value
Input:	0.001 to 99999.9999, max. 4 decimal places

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

iTNC 530:	6540.0

safetyDistStylus   114204

Safety zone around the stylus for pre-positioning

Path:	System ► ProbeSettings ► CfgTTRoundStylus ► [Key name of the TT] ► safetyDistStylus
Available as of:	597110-03
Format:	Numerical value
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
	Safety clearance in the plane perpendicular to the tool axis

Default:	5 mm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

iTNC 530:	6540.1

CfgTTRectStylus 114300

Configuration of a rectangular stylus

Path:	System ► ProbeSettings ► CfgTTRectStylus		
Available as of:	597110-15		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

centerPos 114313

Coordinates of the stylus center

Path:	System ► ProbeSettings ► CfgTTRectStylus ► [Key name of the TT] ► centerPos		
Available as of:	597110-15		
Format:	Numerical value		
Input:	Coordinates of the stylus center with respect to the machine datum -99999.9999 to 99999.9999 [mm], max. 4 decimal places		
Default:	0 mm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-15		
Format:	Numerical value		
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places Safety clearance in tool axis direction		
Default:	10 mm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6540.0		

safetyDistStylus 114318

Safety zone around the stylus for pre-positioning

Path:	System ► ProbeSettings ► CfgTTRectStylus ► [Key name of the TT] ► safetyDistStylus
Available as of:	597110-15
Format:	Numerical value
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
Default:	5 mm	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	6540.1

ChannelSettings

CfgActivateKinem			204000
Active kinematics			
Path:	Channels ► ChannelSettings ► CfgActivateKinem		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

kinemToActivate			204001
Kinematics to be activated / active kinematics			
Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgActivateKinem ► kinemToActivate		
Available as of:	597110-03		
Format:	String		
Input:	max. 18 Characters		
	Key names from channels/kinematics/**CfgKinComposModel**. Select the key name of the kinematic model to be activated. You can also read the currently active kinematic model from this machine parameter.		
Default:		Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

kinemAtStartup			204002
The kinematics to be activated during control start-up			
Path:	Channels ► ChannelSettings ► CfgActivateKinem ► [Key name of the machining channel] ► kinemAtStartup		
Available as of:	597110-06		
Format:	String		
Input:	max. 18 Characters		
	Enter here the key name of a default kinematic model (from **CfgKinComposModel**), that is activated during every control start-up (independently of which key name is entered in the machine parameter **kinemToActivate** (204001)).		
Default:		Optional:	✓
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

iTNC 530:   7506

CfgNcPgmBehaviour   200800

Specify the behavior of the NC program.

Path:	Channels ► ChannelSettings ► CfgNcPgmBehaviour	
Available as of:	597110-0	
Format:	List	
Data object:		
Default:		Optional: -
Reaction:	Program run is locked (RUN)	Access: LEVEL2

operatingTimeReset   200801

Reset the machining time when program starts.

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgNcPgmBehaviour ► operatingTimeReset	
Available as of:	597110-01	
Format:	Selection menu	
Input:	**TRUE** The machining time is reset at each program start. **FALSE** The machining time is totaled.	
Default:	FALSE	Optional: -
Reaction:	Shut down the control (RESET)	Access: LEVEL2

plcSignalCycle   200803

PLC signal for the number of the pending machining cycle

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgNcPgmBehaviour ► plcSignalCycle	
Available as of:	597110-09	
Format:	String	
Input:	max. 500 Characters Name or number of a PLC word marker	
Default:	No value	Optional: ✓
Reaction:	Program run is locked (RUN)	Access: LEVEL2

plcSignalCycState   200805

PLC signal for type of current cycle execution

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgNcPgmBehaviour ► plcSignalCycState		
Available as of:	597110-18		
Format:	Numerical value		
Input:	The following value is written to the configured operand: ■ 0: No machining cycle is being executed ■ 1: Pre-positioning ■ 2: Machining		
Default:		Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL2

CfgGeoTolerance   200900

Geometry tolerances

Path:	Channels ► ChannelSettings ► CfgGeoTolerance		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

circleDeviation   200901

Permissible deviation of the radius

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoTolerance ► circleDeviation		
Available as of:	597110-01		
Format:	Numerical value		
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.		
Default:	0.005 mm	Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1
iTNC 530:	7431		

threadTolerance   200902

Permissible deviation in successive threads

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoTolerance ► threadTolerance		

Available as of:	597110-11		
Format:	Numerical value		
Input:	0.0001 to 999.9999 [mm], max. 9 decimal places		
	Permissible deviation of the dynamically smoothed contour from the programmed thread contour.		
Default:	0.1 mm	Optional:	✓
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

moveBack   200903

Reserve for retraction movements

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoTolerance ► moveBack		
Available as of:	597110-14		
Format:	Numerical value		
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.		
Default:	No value mm	Optional:	✓
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

CfgGeoCycle   201000

Configuration of the fixed cycles

Path:	Channels ► ChannelSettings ► CfgGeoCycle		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

pocketOverlap   201001

Overlap factor for pocket milling

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► pocketOverlap		
Available as of:	597110-01		
Format:	Numerical value		
Input:	0.001 to 1.414, max. 3 decimal places		
Default:	1	Optional:	-

Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7430		

posAfterContPocket   201007

Traverse after machining the contour pocket

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► posAfterContPocket
Available as of:	597110-09
Format:	Selection
Input:	**PosBeforeMachining** Move to the position from which the SL cycle was started. **ToolAxClearanceHeight** Move the tool axis to clearance height.

Default:	PosBeforeMachining	Optional:	✓
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1
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
Available as of:	597110-01
Format:	Selection menu
Input:	**on** The error message is displayed **off** The error message is not displayed

Default:	on	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7441		

displayDepthErr   201003

Display the **Check the depth sign** error message

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► displayDepthErr
Available as of:	597110-01
Format:	Selection menu
Input:	**on**

Error message is displayed

off

Error message is not displayed

Default:	on	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
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
Available as of:	597110-01
Format:	Selection
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.

Default:	CircleTangential	Optional:	-
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1
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
Available as of:	597110-01
Format:	Numerical value
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.

Default:	0	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

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		
Available as of:	597110-05		
Format:	Selection		
Input:	**on** Error message is not displayed **off** Error message is displayed		
Default:	off	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

---

restoreCoolant   201008

Behavior of M7 and M8 with Cycles 202 and 204

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► restoreCoolant		
Available as of:	597110-11		
Format:	Selection menu		
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.		
Default:	TRUE	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7682		

---

facMinFeedTurnSMAX   201009

Automatic feed rate reduction after attaining SMAX

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► facMinFeedTurnSMAX
Available as of:	597110-11
Format:	Numerical value
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 cutting speed (VCONST:ON).

The machine parameter specifies whether the feed rate should be automatically reduced from this point up to the center of rotation.

Settings options:

- 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 F_{min} is:
  F_{min} = feed rate from turning cycle * factor

Default:	100 %	Optional:	✓
Reaction:	Allowed in strobe (SYNC)	Access:	LEVEL1

---

suppressResMatlWar   201010

Do not show "Residual material" warning

Path:	Channels ► ChannelSettings ► [Key name of the machining channel] ► CfgGeoCycle ► suppressResMatlWar
Available as of:	597110-12
Format:	Selection
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.

Default:	Never	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgThreadSpindle   113600

Special spindle parameters for threads

Path:	Channels ► ChannelSettings ► CfgThreadSpindle
Available as of:	597110-0
Format:	List
Data object:	

Default:		Optional:	-

Reaction:	Program run is locked (RUN)	Access:	LEVEL1

sourceOverride   113603

Effective override potentiometer for feed rate during thread cutting

Path:	Channels ► ChannelSettings ► [Key name of machining channel] ► CfgThreadSpindle ► sourceOverride
Available as of:	597110-05
Format:	Selection
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.

Default:		Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

thrdWaitingTime   113601

Waiting time at reversal point in thread base

Path:	Channels ► ChannelSettings ► [Key name of machining channel] ► CfgThreadSpindle ► thrdWaitingTime
Available as of:	597110-03
Format:	Numerical value
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.

Default:		Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7120.0		

thrdPreSwitchTime   113602

Advanced switching time of spindle

Path:	Channels ► ChannelSettings ► [Key name of machining channel] ► CfgThreadSpindle ► thrdPreSwitchTime

Available as of:	597110-03
Format:	Numerical value
Input:	0 to 1 000 [s], max. 9 decimal places The spindle is stopped at this time before reaching the bottom of the thread.

Default:		Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
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
Available as of:	597110-06
Format:	Selection menu
Input:	**TRUE** Spindle speed is limited so that it runs with constant speed approx. 1/3 of the time **FALSE** Limit not active

Default:	FALSE	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1
iTNC 530:	7160, Bit1		

CfgEditorSettings

CfgEditorSettings			105400
Settings for the NC editor			
Path:	System ► EditorSettings ► CfgEditorSettings		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

createBackup			105401
Generate a backup file *.bak			
Path:	System ► EditorSettings ► CfgEditorSettings ► createBackup		
Available as of:	597110-02		
Format:	Selection menu		
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.		
Default:	TRUE	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

deleteBack			105402
Behavior of the cursor after deletion of lines			
Path:	System ► EditorSettings ► CfgEditorSettings ► deleteBack		
Available as of:	597110-02		
Format:	Selection menu		
Input:	**TRUE** Behavior as with iTNC 530, the cursor is on the previous line **FALSE** The cursor is on the next line		
Default:	TRUE	Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

lineBreak   105404

Line break on NC blocks with more than one line

Path:	System ► EditorSettings ► CfgEditorSettings ► lineBreak
Available as of:	597110-02
Format:	Selection
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

Default:	ALL	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7281.0		

stdTNChelp   105405

Activate help graphics when entering cycle data

Path:	System ► EditorSettings ► CfgEditorSettings ► stdTNChelp
Available as of:	597110-01
Format:	Selection menu
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.

Default:	TRUE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

warningAtDEL   105407

Confirmation request when deleting an NC block.

Path:	System ► EditorSettings ► CfgEditorSettings ► warningAtDEL
Available as of:	597110-01
Format:	Selection menu

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.		
Default:	No value, same behavior as for FALSE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
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		
Available as of:	597110-02		
Format:	Numerical value		
Input:	The available value range depends on the performance of the control. For the TNC7 basic, you can enter a value between 100 and .		
	If the parameter is not part of the configuration, the minimal value 100 becomes effective.		
Default:	No value	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7229		

blockIncrement   105409

ISO programming: Block number increment

Path:	System ► EditorSettings ► CfgEditorSettings ► blockIncrement		
Available as of:	597110-04		
Format:	Numerical value		
Input:	0 to 250		
Default:	10	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7220		

useProgAxes   105410

Specify programmable axes

Path:	System ► EditorSettings ► CfgEditorSettings ► useProgAxes
Available as of:	597110-01
Format:	Selection menu
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.

Default:	FALSE	Optional:	✓
Reaction:	Shut down the control (RESET)	Access:	LEVEL1

enableStraightCut   105411

Allow or lock paraxial positioning blocks

Path:	System ► EditorSettings ► CfgEditorSettings ► enableStraightCut
Available as of:	597110-0
Format:	Selection menu
Input:	**TRUE**
	Paraxial positioning blocks are allowed. When an orange axis key is pressed, and in 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 basic generates a straight-line interpolation (L block) instead of a paraxial positioning block.

Default:	No value, behavior as for TRUE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1
iTNC 530:	7246		

noParaxMode   105413

Hide **FUNCTION PARAXCOMP/PARAXMODE**

Path:	System ► EditorSettings ► CfgEditorSettings ► noParaxMode
Available as of:	597110-11
Format:	Selection menu
Input:	Use **noParaxMode** (105413) to hide the **FUNCTION PARAXCOMP** and **FUNCTION PARAXMODE** functions.

FALSE

The functions are displayed

TRUE

The functions are not displayed

If the optional machine parameter does not exist in the configuration, the system behaves as if it were set to **FALSE**.

Default:	–	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

quotePaths			105414
Put all path information in quotation marks			
Path:	System ► EditorSettings ► CfgEditorSettings ► quotePaths		
Available as of:	597110-18		
Format:	Selection menu		
Input:	**TRUE** Path information is enclosed in quotation marks. **FALSE** Path information is not enclosed in quotation marks.		
Default:		Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgPgmMgt

CfgPgmMgt			122100
Settings for the file management			
Path:	System ► ProgramManager ► CfgPgmMgt		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL1

CfgProgramCheck

CfgProgramCheck		129800
Settings for tool-usage files		

Path:	System ► ToolSettings ► CfgProgramCheck	
Available as of:	597110-14	
Format:	List	
Data object:		
Default:		Optional: -
Reaction:	Program run is locked (RUN)	Access: LEVEL1

autoCheckTimeOut		129803
Timeout for creation of tool-usage files		

Path:	System ► ToolSettings ► CfgProgramCheck ► autoCheckTimeOut	
Available as of:	597110-15	
Format:	Numerical value	
Input:	Automatic creation of the tool-usage file is aborted if this time is exceeded. 1 to 500	
Default:	30 min	Optional: ✓
Reaction:	Program run is locked (RUN)	Access: LEVEL1

autoCheckPrg		129801
Create tool-usage file for NC program		

Path:	System ► ToolSettings ► CfgProgramCheck ► autoCheckPrg	
Available as of:	597110-14	
Format:	Numerical value	
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 by using an editor.	
Default:		Optional: ✓

| Reaction: | Program run is locked (RUN) | Access: | LEVEL1 |

autoCheckPal			129802
Create pallet-usage files			
Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckPal		
Available as of:	597110-14		
Format:	Numerical value		
Input:	**NoAutoCreate**		
	No tool-usage files will be generated upon pallet selection.		
	OnProgSelectionIfNotExist		
	Upon pallet selection, tool-usage lists that do not already exist will be generated.		
	OnProgSelectionIfNecessary		
	Upon pallet selection, 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 using an editor.		
Default:	OnProgSelectionAndModify	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgUserPath

CfgUserPath				102200
Paths for the end user				
	Path:	System ► Paths ► CfgUserPath		
	Available as of:	597110-0		
	Format:	List		
	Data object:			
	Default:		Optional:	-
	Reaction:	Program run is locked (RUN)	Access:	LEVEL1

ncDir				102201
List of drives and/or directories				
	Path:	System ► Paths ► CfgUserPath ► ncDir		
	Available as of:	597110-01		
	Format:	String		
	Input:	max. 260 Characters		
		This parameter is available only on the Windows programming stations of the TNC7 basic. 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.		
	Default:	TNC:\	Optional:	-
	Reaction:	Program run is locked (RUN)	Access:	LEVEL1

fn16DefaultPath				102202
Default output path for the **FN 16: F-PRINT** function in the Program Run operating modes				
	Path:	System ► Paths ► CfgUserPath ► fn16DefaultPath		
	Available as of:	597110-05		
	Format:	String		
	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.		
	Default:	TNC:\	Optional:	✓

Reaction:	Program run is locked (RUN)	Access:	LEVEL1

fn16DefaultPathSim 102203

Default output path for the **FN 16: F-PRINT** function in the Programming and Test Run operating modes

Path:	System ▶ Paths ▶ CfgUserPath ▶ fn16DefaultPathSim		
Available as of:	597110-05		
Format:	String		
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 lands in the directory specified here.		
Default:	TNC:\	Optional:	✓
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

serialInterfaceRS232

CfgSerialPorts   106600

Data record belonging to the serial port

Path:	System ► Network ► Serial ► CfgSerialPorts		
Available as of:	597110-0		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

activeRs232   106601

Enable the RS-232 interface in the program manager

Path:	System ► Network ► Serial ► CfgSerialPorts ► activeRs232		
Available as of:	597110-03		
Format:	Selection menu		
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.		
Default:	No value	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

baudRateLsv2   106606

Data transfer rate for LSV2 communication in baud

Path:	System ► Network ► Serial ► CfgSerialPorts ► baudRateLsv2
Available as of:	597110-01
Format:	Selection
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

Default:	BAUD_115200	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

CfgSerialInterface 106700

Definition of data records for the serial ports

Path:	System ► Network ► Serial ► CfgSerialInterface
Available as of:	597110-0
Format:	List
Data object:	

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

baudRate 106701

Data transfer rate for communication in baud

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► baudRate
Available as of:	597110-01
Format:	Selection
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

Default:	BAUD_57600	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5040		

protocol 106702

Communications protocol

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► protocol
Available as of:	597110-01
Format:	Selection
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.

Default:	BLOCKWISE	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5030		

dataBits 106703

Data bits in each transferred character

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► dataBits
Available as of:	597110-01
Format:	Selection
Input:	**7 bits** 7 data bits are transferred for each character transferred. **8 bits** 8 data bits are transferred for each character transferred.

Default:	7 bits	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

iTNC 530:	5020 Bit0

parity 106704

Type of parity checking

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► parity		
Available as of:	597110-01		
Format:	Selection		
Input:	**NONE** No parity **EVEN** Even parity **ODD** Odd parity		
Default:	EVEN	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5020 Bit4/5		

stopBits 106705

Number of stop bits

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► stopBits		
Available as of:	597110-01		
Format:	Selection		
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.		
Default:	1 stop bit	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5020 Bit6/7		

flowControl 106706

Type of data-flow checking

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► flowControl
Available as of:	597110-01
Format:	Selection

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

Default:	RTS_CTS	Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
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
Available as of:	597110-01
Format:	Selection
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.

Default:	FE1	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

bccAvoidCtrlChar   106708

Avoid control characters in the block check character (BCC)

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► bccAvoidCtrlChar
Available as of:	597110-01
Format:	Selection menu
Input:	**TRUE**
	Ensures that the check sum does not correspond to a control character
	FALSE

	Function not active		
Default:	FALSE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5020 Bit1		

rtsLow 106709

Idle state of the RTS line

Path:	System ► Network ► Serial ► CfgSerialInterface ► [Key names of the interface parameters] ► rtsLow		
Available as of:	597110-03		
Format:	Selection menu		
Input:	**TRUE**		
	The idle state of the RTS line is logical LOW		
	FALSE		
	The idle state of the RTS line is at logical HIGH		
Default:	FALSE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
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		
Available as of:	597110-01		
Format:	Selection menu		
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.		
Default:	FALSE	Optional:	✓
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2
iTNC 530:	5020 Bit9		

Monitoring

CfgCompMonUser   129400

User settings for component monitoring

Path:	System ► Monitoring ► CfgCompMonUser		
Available as of:	597110-14		
Format:	List		
Data object:			
Default:		Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

enforceReaction   129401

The configured error reactions are enforced

Path:	System ► Monitoring ► CfgCompMonUser ► enforceReaction		
Available as of:	597110-13		
Format:	Selection menu		
Input:	**TRUE**		
	FALSE		
Default:	TRUE	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

showWarning   129402

Display warnings of monitoring tasks

Path:	System ► Monitoring ► CfgCompMonUser ► showWarning		
Available as of:	597110-13		
Format:	Selection menu		
Input:	**TRUE**		
	FALSE		
Default:	TRUE	Optional:	-
Reaction:	Program run is locked (RUN)	Access:	LEVEL1

CfgMachineInfo

CfgMachineInfo				131700
General information of the machine operator				
Path:	System ► CfgMachineInfo			
Available as of:	597110-14			
Format:	List			
Data object:	Defines general information about this machine: ■ Settable by the user of the machine ■ Can be queried (e.g., via the OPC UA NC Server)			
Default:		Optional:	-	
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2	

machineNickname				131701
Custom name (nickname) of the machine				
Path:	System ► CfgMachineInfo ► machineNickname			
Available as of:	597110-14			
Format:	String			
Input:	max. 64 Characters Machine designation freely selectable by the user.			
Default:		Optional:	-	
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2	

inventoryNumber				131702
Inventory number or ID				
Path:	System ► CfgMachineInfo ► inventoryNumber			
Available as of:	597110-14			
Format:	String			
Input:	max. 64 Characters Internal inventory number of the operator's machine.			
Default:		Optional:	-	
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2	

image				131703
Photo or image of the machine				
Path:	System ► CfgMachineInfo ► image			

Available as of:	597110-14
Format:	String
Input:	max. 260 Characters
	Path to an image file (*.jpg or *.png).

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

location   131704

Machine location

Path:	System ► CfgMachineInfo ► location
Available as of:	597110-14
Format:	String
Input:	max. 64 Characters

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

department   131705

Department or division

Path:	System ► CfgMachineInfo ► department
Available as of:	597110-14
Format:	String
Input:	max. 64 Characters

Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

responsibility   131706

Responsible for the machine

Path:	System ► CfgMachineInfo ► responsibility
Available as of:	597110-14
Format:	String
Input:	max. 64 Characters
	Contact partner responsible for the machine, can be a person or a department.

Default:		Optional:	-

Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

contactEmail   131707

Contact email address

Path:	System ► CfgMachineInfo ► contactEmail		
Available as of:	597110-14		
Format:	String		
Input:	max. 64 Characters		
	E-mail address of the responsible person or department.		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

contactPhoneNumber   131708

Contact phone number

Path:	System ► CfgMachineInfo ► contactPhoneNumber		
Available as of:	597110-14		
Format:	String		
Input:	max. 32 Characters		
	Telephone number of the responsible person or department.		
Default:		Optional:	-
Reaction:	Change at any time (NOTHING)	Access:	LEVEL2

46.3 User administration roles and rights

46.3.1 List of roles

> ℹ The following contents can change in the following software versions of the control:
> - HEROS role names
> - Unix groups
> - Basic ID number

Further information: "Roles", Page 1844

Operating system roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 335
	■ HEROS.Printer	■ lp	■ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	■ HEROS.SetShares	■ mntcfg	■ 334
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 337
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 341
	■ HEROS.SetNetwork	■ netadmin	■ 336
	■ HEROS.SetTimezone	■ tz	■ 333
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.LegacyUserNoCtrlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
	▪ HEROS.BackupMachine	▪ backup	▪ 338
	▪ HEROS.UserAdmin	▪ useradmin	▪ 339

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	▪ NC.OPModeProgramRun	▪ NCOpPgmRun	▪ 302
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	▪ NC.EditNCProgram	▪ NCEdNCProg	▪ 305
	▪ NC.EditPalletTable	▪ NCEdPal	▪ 309
	▪ NC.EditPresetTable	▪ NCEdPreset	▪ 308
	▪ NC.EditToolTable	▪ NCEdTool	▪ 306
	▪ NC.OPModeMDi	▪ NCOpMDI	▪ 301
	▪ NC.OPModeManual	▪ NCOpManual	▪ 300
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the Programmer role, as well as the following rights:		
	▪ NC.ApproveFsAxis	▪ NCApproveFsAxis	▪ 319
	▪ NC.EditPocketTable	▪ NCEdPocket	▪ 307
	▪ NC.SetupDrive	▪ NCSetupDrv	▪ 315
	▪ NC.SetupProgramRun	▪ NCSetupPgRun	▪ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	▪ NC.ScheduleProgramRun	▪ NCSchedulePgRun	▪ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors. ▪ Special functions of Q parameter programming and editing the table header Replacement for code number **555343**		
	▪ NC.EditNCProgramAdv	▪ NCEditNCPgmAdv	▪ 327
	▪ NC.EditTableAdv	▪ NCEditTableAdv	▪ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
	■ NC.RemoteProgramRun	■ NCRe-motePgmRun	■ 329

Machine manufacturer (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	■ NC.ConfigUserAdv ■ NC.SetupDrive	■ NCConfi-gUserAdv ■ NCSetupDrv	■ 316 ■ 315
PLC.ServiceRead	This role allows read-only access during servicing. This role can be used to display various types of diagnostic information		
	■ NC.Data.AccessServiceRead	■ NCDASer-viceRead	■ 324

⚙ Refer to your machine manual.

The machine manufacturer can adapt the PLC roles.

When the **Machine manufacturer (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:

- The names of the roles
- The number of roles
- The functionality of the roles

46.3.2 List of rights

The table below lists all of the individual rights.

Further information: "Rights", Page 1844

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operating the machine in the **Manual Operation** and **Electronic handwheel** operating modes.
NC.OPModeMDi	Working in the **Positioning w/ Manual Data Input** operating mode.
NC.OpModePro-gramRun	Execution of NC programs in the **Program Run Full Sequence** or **Program run, single block** operating mode.
NC.SetupProgram-Run	Probing in the **Manual Operation** and **Electronic handwheel** operating modes. Using the **AFC** and **ACC** functions.
NC.SchedulePro-gramRun	Programming a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes
NC.EditNCProgra-mAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Adjustment of date and time, time zone and time synchro-nization via NTP and the **HEROS menu**.
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.Moun-tShares	Connecting and disconnecting network shares with the control
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMa-chine	Backup and restoring data of the entire machine configura-tion
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users

HEROS role name	Description
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	NC program start from an external application (e.g., via the DNC interface)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.DataAccessServiceRead	Read-only access to the **PLC:** drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through OPC UA NC Server to data defined by the machine manufacturer

46.4 Special functions defining the machine behavior

With code number 555343, you can enable NC functions that are intended for HEIDENHAIN, the machine manufacturer, and third-party providers only.

The following NC functions influence the machine behavior:

- Kinematics functions:
  - **WRITE KINEMATICS**
  - **READ KINEMATICS**
- PLC functions:
  - **FUNCTION SCOPE**
    - **START**
    - **STORE**
    - **STOP**
  - **READ FROM PLC**
  - **WRITE TO PLC**
  - **WRITE CFG**
    - **PREPARE**
    - **COMMIT TO DISK**
    - **COMMIT TO MEMORY**
    - **DISCARD PREPARATION**
- Variable programming:
  - **FN 19: PLC**
  - **FN 20: WAIT FOR**
  - **FN 29: PLC**
  - **FN 37: EXPORT**
- **CYCL QUERY**

NOTICE

Caution: Significant property damage!

The use of special functions for machine behavior might result in undesired behavior and severe errors (e.g., the control might not be operable any longer). With these NC functions, HEIDENHAIN, the machine manufacturer, and third-party providers have the possibility of modifying the machine behavior under program control. It is not recommended that machine operators or NC programmers use this function. There is a danger of collision during the execution of these NC functions and during the subsequent machining operations!

- ► Only use special functions for machine behavior after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider
- ► Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

---

46.5 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 1027
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	3D ROT not permitted
1076	Activate 3D 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	3D 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

46.6 System data

46.6.1 List of FN functions

The **FN 18: SYSREAD** function can be used to read numeric system data and save the value in a Q, QL, or QR parameter (e.g., **FN 18: SYSREAD Q25 = ID210 NR4 IDX3**.)

> ℹ The control always outputs system data in the metric system with **FN 18: SYSREAD**, regardless of the unit of the NC program.

The **SYSSTR** function can be used to read alphanumeric system data and save the value in a QS parameter (e.g., **QS25 = SYSSTR(ID 10950 NR1)**).

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
			-	Within an M function macro: Number of the M function. Otherwise −1
		10	-	Repeat counter: Indicates the number of times the current code has been executed since the current NC program call

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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.
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	-	Number or name (string or QS) of the label to which the NC program will jump if **FN 14: ERROR** has been programmed with the NC CANCEL reaction, instead of aborting the NC program with an error message. The error number programmed in the **FN 14** command can be read under ID992 NR14. Value = 0: **FN 14** has a 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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		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
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		48	-	Tolerance
		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
		70	-	Multiplier for feed rate (cycles 17 and 18)
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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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)
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 rows in a freely definable table that has been opened with **FN 26: 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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		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 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
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize	Active radius

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			and oversize from TOOL CALL	
		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 to 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 to 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 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,

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				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)
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 to 3 (rotA, rotB, rotC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read OEM offset values.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 to 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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
Nominal positions of the physical axes in the REF system				
	245	1	Axis	Current nominal positions of the physical axes 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 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
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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:
		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 to 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)
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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		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 Global Program Settings active 1 = Any GPS settings active

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No Global Program Settings active 1 = Any GPS settings 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
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 to 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 to 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 to 10 (X, Y, Z, A, B, C, U, V, W, VT)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		9	Axis	GPS: Value for handwheel superimposi-tioning Index: 1 to 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 to 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 to 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
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
			3	TT: Designation of the active line in the touch-probe table
			4	TT: Touch probe input
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			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
		100	-	Distance after which the probe is deflected during touch probe simulation
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 3D 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
Preset from the touch probe cycle (probing results)				
	360	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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Preset from touch probe cycle (probing results)				
	360	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
Settings for touch probe cycles				
	370	2	-	Rapid traverse for measurement
		3	-	Machine rapid traverse as rapid traverse for measurement
		5	-	Angle tracking on/off
		6	-	Automatic measuring cycles: interruption with info about on/off
Settings for touch-probe cycles				
	370	7	-	Reaction when the automatic 14xx measuring cycle does not reach the probing point: 0 = Cancellation 1 = Warning 2 = No message In case of values 1 and 2, the measurement result must be evaluated, and a corresponding reaction is required.
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 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 to 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
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, then the function returns the value -1.
		2	-	Number of the active pallet preset. Same as NO1.
Values for the basic transformation of the pallet preset				
	547	Row number	Axis	Read the basic transformation values 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 the axis-offset values from the pallet preset table.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset.. Index: 4 to 9 (A_OFFS, B_OFFS, C_OFFS,...)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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_min-CornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPath-Jerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPath-JerkHi**) in m/s^3
		6	-	Tolerance at low speeds (**MP_pathToler-ance**) in mm
		7	-	Tolerance at high speeds (**MP_pathToler-anceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPa-thYank**) 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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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**)
		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**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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**)
		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
		49	-	Filter reduction mode (**CfgPositionFilter/timeGainAtStop**) 0 = Off 1 = NoOvershoot 2 = FullReduction
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_lpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)
		53	Index of physical axis	Radial jerk, normal feed rate (**MP_maxTransJerk**)
		54	Index of physical axis	Radial jerk, high feed rate (**MP_maxTransJerkHi**)
Read or write look-ahead parameters of a single axis (at cycle level)				
	613	see ID610	see ID610	Same as ID610 but is only in effect at the cycle level. Overwrite values from the machine configuration and values at the machine level. **Further information:** "FN functions ID610, ID611, ID613", Page

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
		3	-	Read the SIK type (generation) 1 = SIK1 or no SIK 2 = SIK2
		4	Option number (4 digits)	Read the status of a software option (only available with SIK2) 0 = Not enabled 1 or higher = Number of enabled options
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC7, TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
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
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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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)
		48	-	Radius at the tool tip (R_TIP)
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 NO1
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. 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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				–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)
		28	-	Read inclination angle of the current tool spindle
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 **FN 14** error
		16	-	Real execution active? 1 = execution, 0 = simulation

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		17	-	2D 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
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	Cycle number	Single cycle enabled: 0 = No 1 = Yes
		33	-	Write-access enabled for DNC (Python scripts) for executed entries in the pallet table: 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
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Configuration settings for cycles				
	1030	1	-	Display the **Spindle is not rotating** error message (CfgGeoCycle/**displaySpindleErr**) 0 = No, 1 = Yes
		2	-	Display the **Check the depth sign** 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 0 = Circular interpolation test 1 = Waterfall chart test 2 = Frequency response 3 = Envelope curve spectrum 4 = Advanced frequency response
			2	Component monitoring: Index of the axis from CfgAxes**axisList**
			3 – 9	Component monitoring: further arguments depend on the measurement
		2	3 – 9	Component monitoring: further arguments depend on the measurement
		3	0	KinematicsOpt: Read the current cycle number (450-453)
		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
Program information (system string)				
	10010	1	0/1/2/3	IDX0 = Complete path of the current main program of pallet program IDX1 = File path of the directory where the NC program is located IDX2 = Name of the NC program, without the path and file extension IDX3 = File extension of the NC program
		2	0/1/2/3	IDX0 = Complete path of the NC program visible in the block display IDX1 = File path of the directory where the NC program is located IDX2 = Name of the NC program without the path and file extension IDX3 = File extension of the NC program
Read program information (system string)				
	10010	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
		13	-	Symbolic name of the grinding tool table
		14	-	Symbolic name of the dressing tool table
		21	-	Symbolic name of the compensation table in the T-CS tool coordinate system

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		22	-	Symbolic name of the compensation table in the WPL-CS working plane coordinate system
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	0 - 16, 20	-	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**)
		51	-	Shape of stylus from column STYLUS in 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**.
		74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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)				
	10630	10	-	The string corresponds to the format of the version ID shown (e.g., **340590 09** or **817601 05 SP1**)
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 current tool data (system string)				
	10950	6	-	Entry from the TSHAPE column - file name of the 3D tool shape (*.stl)
Read information from OEM macros and HEIDENHAIN cycles (system string)				
	11031	10	-	Returns the selection of the FUNCTION MODE SET <OEM mode> macro as a string.
		100	-	Cycle 238: list of key names for component monitoring
		101	-	Cycle 238: file names for log file

46.7 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 340 (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 1286911	-02	-03	-04	-05

	⇧
ID 1286914	-03

	⇧	CTRL
ID 1286915	-02	-03

ID 1286917	-01

Keycaps for operating aids

	PGM MGT	↻	ERR	CALC	MOD	HELP
ID 1286909	-61	-62	-63	-64	-65	-66

Keycaps for operating modes

	🖐	⊚	≡▶	⇨	▣▶	▤▶	→▶	→
ID 1286909	-67	-68	-69	-70	-71	-72	-73	-74

Keycaps for programming

	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

Keycaps for axis input and value input

ID 1286909	X Orange -94	Y Orange -95	Z Orange -96	A Orange -4K	B Orange -4Y	C Orange -4L	U Orange -5K	V Orange -98	W Orange -4Z

ID 1286909	IV Orange -97	-0N	-3S	ESC -4S	INS -4T	-3R	i -3T	-3U	DEL -3V

ID 1286909	7 -0B	8 -0C	9 -0D	4 -0E	5 – -03*)	1 -0G	2 -0H	3 -2L	0 -2M
ID 1344337*)	–	–	–	–	–	–	–	–	–

*)   With tactile mark

ID 1286909	. -0K	−/+ -0L	⟨x⟩ -0M	Q -2N	CE -0P	DEL -2P	NO ENT -0R	END -0S	-3N

ID 1286909	>> -3W	-3P	P Orange -99	I Orange -0A					

ID 1286914	ENT -04

Keycaps for navigation

ID 1286909	-0T	HOME -0U	PG UP -0V	-0W	GOTO – -04*)	-0Y	END -0Z	PG DN -1A
ID 1344337*)	–	–	–	–	–	–	–	–

*)   With tactile mark

ID 1344337*)	↑ -06	→ -07

*)   With tactile mark

Keycaps for machine functions

	IV+	Z+	Y+	V+	VI+	X+	⊢		Y−
ID 1286909	-1D	-1E	-1F	-1G	-1H	-1K	-1L	-4X	-1N

	IV−	VI−			FN 1				
ID 1286909	-1P	-1R	-1S	-1T	-1U	-1V	-1W	-1X	-1Y

	FN 2			FN 3			Red	Green	X−
ID 1286909	-1Z	-2A	-2B	-2C	-2D	-2E	-2H	-2K	-2R

		Z−	V−	+	−				
ID 1286909	−	-2T	-2U	-2Z	-3A	-3E	-3F	-3G	-3H
ID 1344337*)	-05*)	−	−	−	−	−	−	−	−

*) With tactile mark

						C+		C−	
ID 1286909	-3L	-3M	-3X	-3Y	-3Z	-4A	-4B	-4C	-4D

	W+	W−	Red	A+	A−	B+	B−	Red	Red
ID 1286909	-4E	-4F	-4H	-4M	-4N	-4P	-4R	-4U	-06

	Green	U−	U+			FN 4	FN 5		
ID 1286909	-07	-5A	-5B	-5C	-5D	-4V	-4W	-5E	-5H

						↑	→	↗	
ID 1286909	-5F	-5G	2Y	-3K	-4G	-2V	-2W	-2X	

	Red	Red
ID 1286909	-2F	-2G

Other keycaps

| ID 1286909 | -01 | -02 | Orange -05 | Green -03 | Red -04 | – | – | – | – |

i If you need keycaps with additional symbols, please contact HEIDENHAIN.

Index

HEIDENHAIN