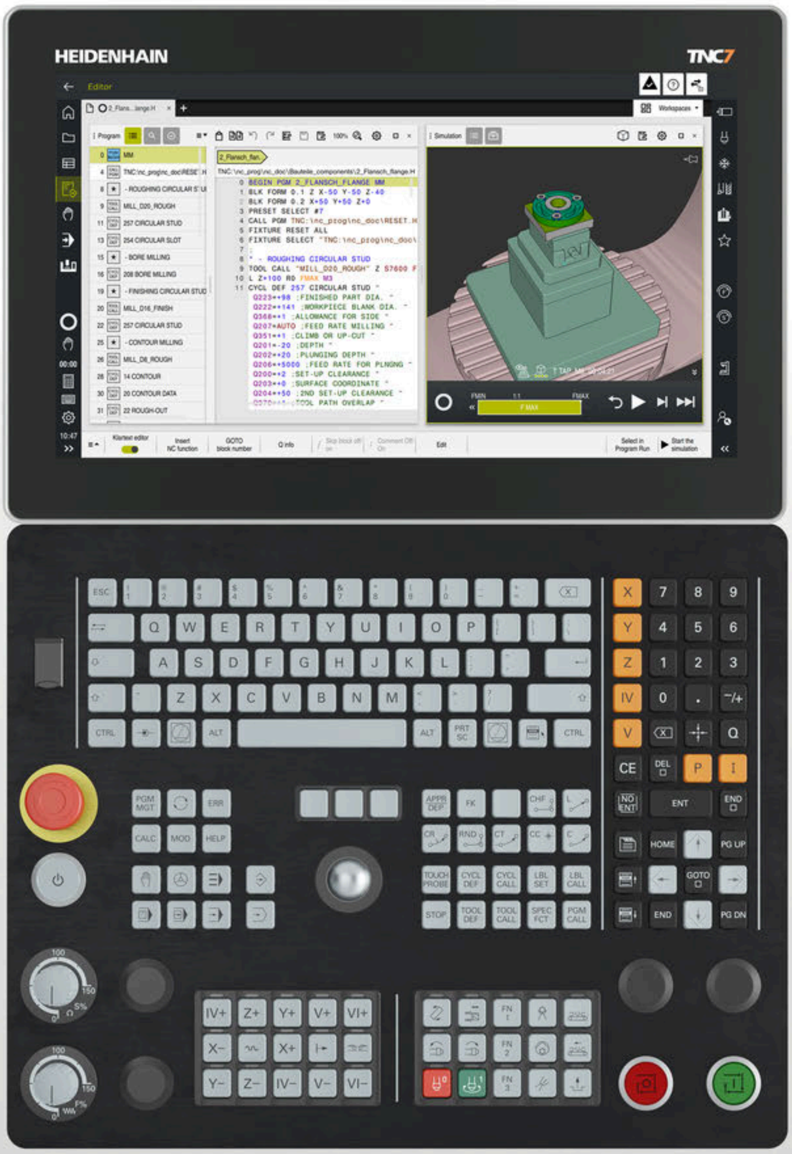
HEIDENHAIN



TNC7 basic

User's Manual
Programming and Testing

NC Software
81762x-18

English (en)
10/2023

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)

> 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 43

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, 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 33

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 43

Related topics

- The **Help** workspace

  Further information: "The Help workspace", Page 602

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.

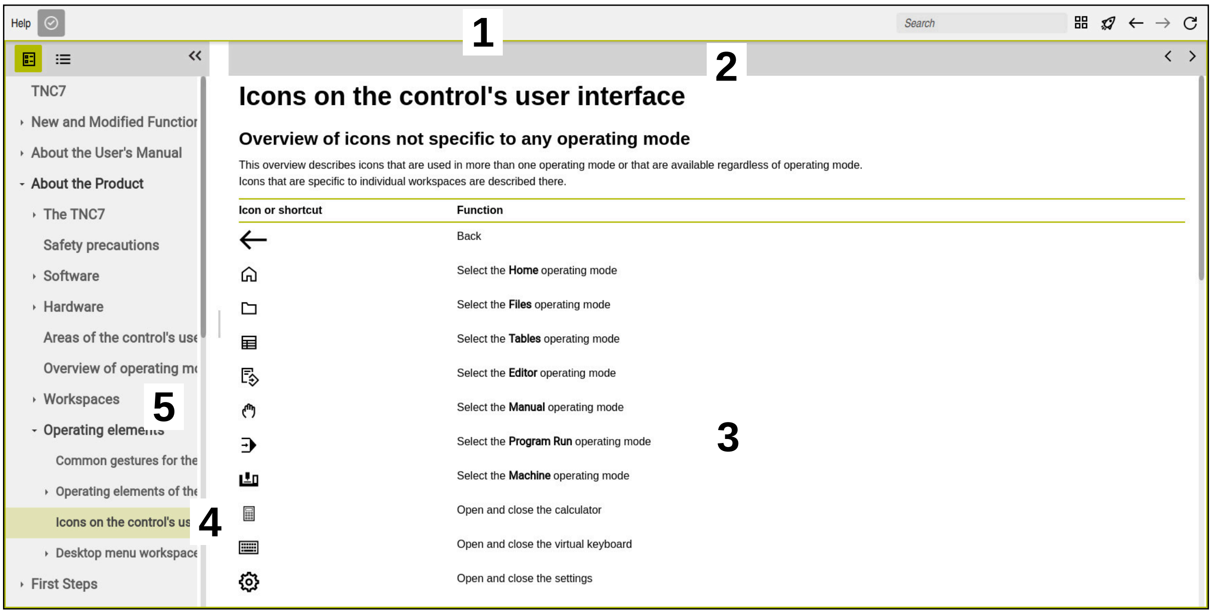
Further information: "The Help application", Page 37

Further information: "The Help workspace", Page 602

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

Further information: "Icons", Page 38

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 38
2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 38
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 39
	**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 40

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 37

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 602

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 35

- 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 629

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: User's Manual for Setup and Program Run

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.

> 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.

> Contribute actively to increasing the safety by reporting any missing or misleading information.
>
> **Further information:** "Contacting the editorial staff", Page 40

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.

> 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! 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
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.

The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 47

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

HEIDENHAIN has simplified the version schema, starting with NC software version 16:

- The publication period determines the version number.
- All control models of a publication period have the same version number.
- The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-18	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: User's Manual for Setup and Program Run

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.

> 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: User's Manual for Setup and Program Run

Overview

> Keep in mind that particular software options also require hardware extensions.
>
> **Further information:** User's Manual for Setup and Program Run

<table>
<tr><th>Software option</th><th>Definition and application</th></tr>
<tr><td>Control Loop Qty.
(#0-3 / #6-01-1*)</td><td>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.</td></tr>
<tr><td>Adv. Function Set 1
(#8 / #1-01-1)</td><td>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 274
■ Programming of contours on a developed cylindrical surface (e.g., with Cycle 27 CYLINDER SURFACE)
Further information: User's Manual for Machining Cycles
■ 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 452
■ 3-axis circular interpolation with a tilted working plane
The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.</td></tr>
<tr><td>Adv. Function Set 2
(#9 / #4-01-1)</td><td>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 315
■ Running of NC programs with vectors, including optional 3D tool compensation
Further information: "3D tool compensation (#9 / #4-01-1)", Page 333
■ Manual moving of axes in the active tool coordinate system T-CS</td></tr>
<tr><td>Touch Probe Function
(#17 / #1-05-1)</td><td>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.
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.</td></tr>
</table>

Software option	Definition and application
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 455 ■ **M118** for superimposed handwheel positioning during program run **Further information:** "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 453 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:** User's Manual for Setup and Program Run
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. **Further information:** User's Manual for Setup and Program Run
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:** User's Manual for Setup and Program Run

Software option	Definition and application
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:** User's Manual for Measuring Cycles for Workpieces and Tools
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:** User's Manual for Setup and Program Run
4 Additional Axes (#77 / #6-01-1*)	**Four additional control loops** **Further information:** "**Control Loop Qty.** (#0-3 / #6-01-1*)", Page 48
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 ■ The **T usage order** shows the tool order of the NC program or pallet to be run **Further information:** User's Manual for Setup and Program Run 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. With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase the efficiency. **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
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 ■ 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 386 **Further information:** User's Manual for Setup and Program Run
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:** User's Manual for Setup and Program Run
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. **Further information:** User's Manual for Setup and Program Run

Software option	Definition and application
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 654
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. **Further information:** User's Manual for Setup and Program Run
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:** User's Manual for Machining Cycles

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
- 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: User's Manual for Setup and Program Run

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 46

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 65

Further information: "Operating elements of the keyboard unit", Page 65

The machine operating panel is machine-dependent.

MB 340 (FS)

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: User's Manual for Setup and Program Run

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.

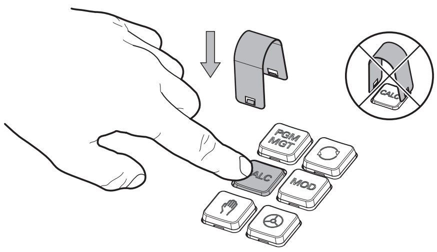
- ▶ Apply a small amount of the cleaning agent onto a cleaning cloth
- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

Exchanging keycaps

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

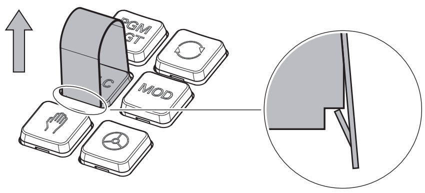
IP54 protection cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

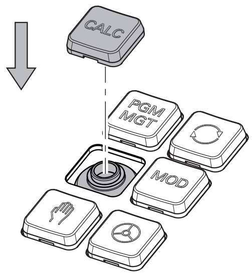


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

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



- Pull off the keycap

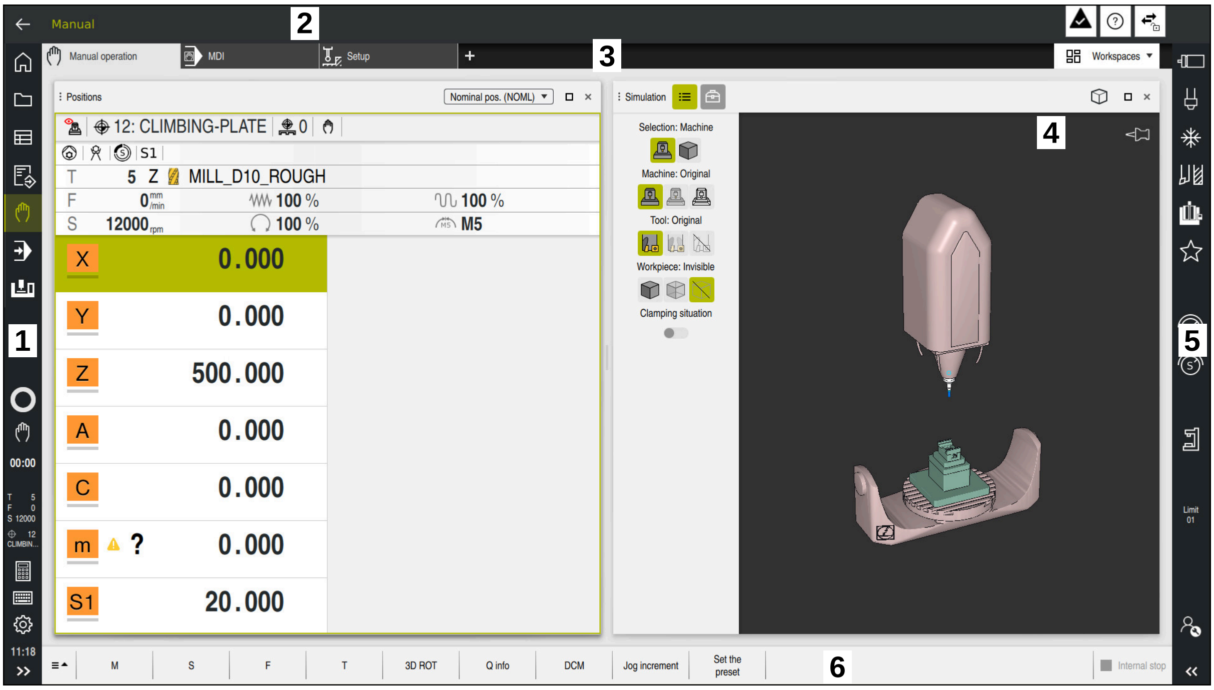


- Place the keycap onto the seal and push it down

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

- Verify proper seating and correct functionality

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 59
- Status overview
  Further information: User's Manual for Setup and Program Run
- Calculator
  Further information: "Calculator", Page 623
- Screen keyboard
  Further information: "Virtual keyboard of the control bar", Page 604
- 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
- **Help** icon for context-sensitive help

  Further information: "Context-sensitive help", Page 39

  Further information: User's Manual for Setup and Program Run
- 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 61

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

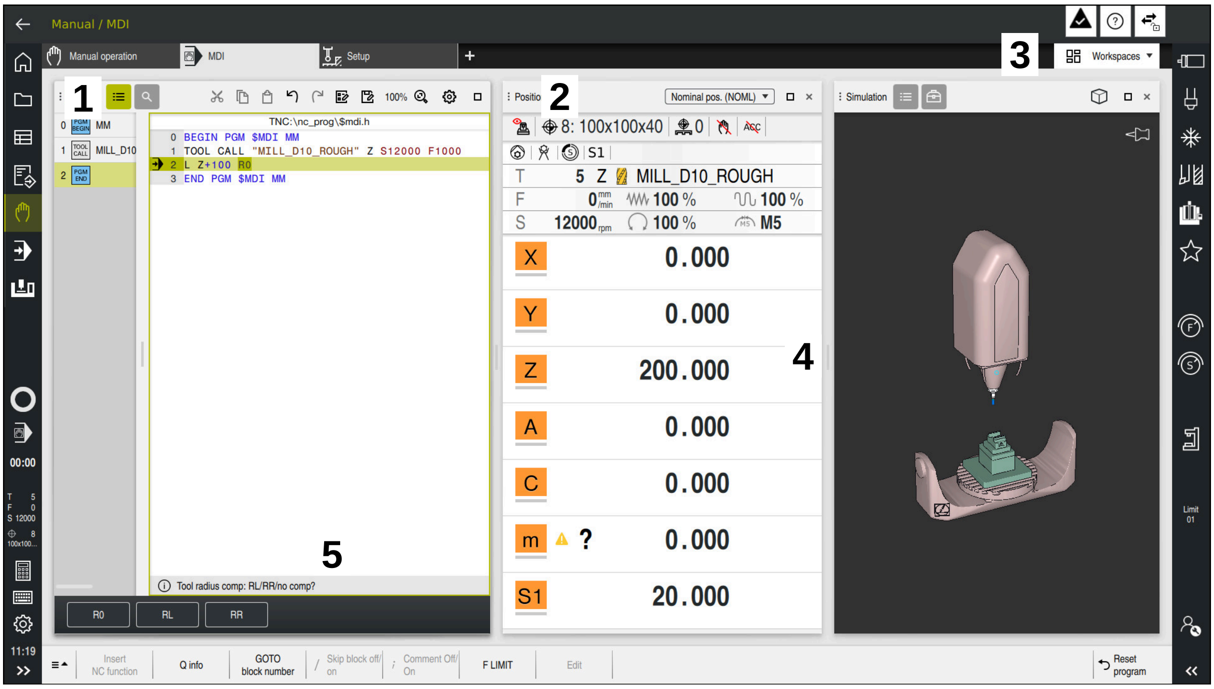
The control provides the following operating modes:

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	See the User's Manual for Setup and Program Run
	■ The **Help** application	Page 602
	■ Applications for machine parameters	See the User's Manual for Setup and Program Run
	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 350
	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 672

Icon	Operating modes	Further information
	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	Page 109
	The **Manual** operating mode contains the following applications:	
	■ The **Manual operation** application	See the User's Manual for Setup and Program Run
	■ The **MDI** Application	See the User's Manual for Setup and Program Run
	■ The **Setup** application	See the User's Manual for Setup and Program Run
	■ The **Move to ref. point** application	See the User's Manual for Setup and Program Run
	■ The **Retract** application You can move the tool away from the workpiece, for example after a power failure.	See the User's Manual for Setup and Program Run
	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.	See the User's Manual for Setup and Program Run
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	See the User's Manual for Setup and Program Run
	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.	See the User's Manual for Setup and Program Run
Job list In the **Job list** workspace, you edit and execute pallet tables.	Page 654
Open File In the **Open File** workspace you select or create files, for example.	Page 360
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 350
Details In the **Details** workspace, the control displays information on the selected machine parameter or the last change you made.	**Further information:** User's Manual for Setup and Program Run
Document You can open files for viewing in the **Document** workspace, for example a technical drawing.	Page 363
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.	See the User's Manual for Setup and Program Run
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 682
The **Form** for pallets In the **Form** workspace the control shows the contents of the pallet table for the selected row.	Page 662

Workspace	Further information
Retract In the **Retract** workspace, you can disengage the tool after a power interruption.	See the User's Manual for Setup and Program Run
Desktop menu In the **Desktop menu** workspace, the control displays selected control and HEROS functions.	Page 76
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 602
Contour graphics 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.	Page 555
List In the **List** workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	See the User's Manual for Setup and Program Run
Positions In the **Positions** workspace, the control displays information about the status of various functions of the control and about current axis positions.	See the User's Manual for Setup and Program Run
Program The control displays the NC program in the **Program** workspace.	Page 110
Referencing On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.	See the User's Manual for Setup and Program Run
Remote Desktop Manager (#133 / #3-01-1) 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.	See the User's Manual for Setup and Program Run
Quick selection 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.	Page 361
Simulation In the **Simulation** workspace, the control shows the simulated or current movements, depending on the operating mode.	Page 629
Simulation status In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	See the User's Manual for Setup and Program Run

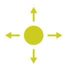
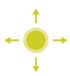
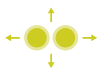
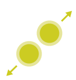
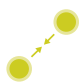
Workspace	Further information
Start/Login In the **Start/Login** workspace, the control shows the steps that are performed during startup.	Page 80
Status In the **Status** workspace, the control shows the status and values of individual functions.	See the User's Manual for Setup and Program Run
Table In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.	Page 676
The **Table** for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	See the User's Manual for Setup and Program Run
Keyboard In the **Keyboard** workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 604
Overview In the **Overview** workspace, the control displays information on the status of individual functional safety (FS) safety functions.	See the User's Manual for Setup and Program Run

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 65

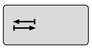
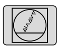
In addition, the control's keyboard unit offers keys and other elements for alternative operating sequences.

Description of function

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

> 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 604

Keycaps for alphabetic keyboard

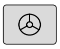
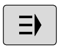
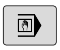
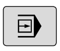
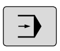
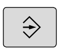
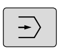
Key	Meaning
A B C	Enter texts (e.g., file names)
Q Q QL QR	**Q** With an open NC program, enter a Q parameter formula in the **Editor** operating mode, or in the **Manual** operating mode open the **Q parameter list** window **Further information:** "The Q parameter list window", Page 484 By selecting 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:** User's Manual for Setup and Program Run
	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 360
	Currently no function
ERR	Open and close the message menu **Further information:** User's Manual for Setup and Program Run
CALC	Open and close the calculator **Further information:** "Calculator", Page 623
MOD	Open the **Settings** application **Further information:** User's Manual for Setup and Program Run
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 36

Operating modes

> 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:** User's Manual for Setup and Program Run
	Activate and deactivate the electronic handwheel in the **Manual** operating mode **Further information:** User's Manual for Setup and Program Run
	Open the **Tool Management** tab in the **Tables** operating mode **Further information:** User's Manual for Setup and Program Run
	Open the **MDI** application in the **Manual** operating mode **Further information:** User's Manual for Setup and Program Run
	Open the **Program Run** operating mode in **Single Block** mode **Further information:** User's Manual for Setup and Program Run
	Open the **Program Run** operating mode **Further information:** User's Manual for Setup and Program Run
	Open the **Editor** operating mode **Further information:** "The Editor operating mode", Page 109
	While the NC program is running, open the **Simulation** workspace in the **Editor** operating mode **Further information:** "The Simulation Workspace", Page 629

Keycaps for NC dialog

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

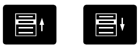
Key	Meaning
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function **Further information:** "Fundamentals of approach and departure functions", Page 192
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode **Further information:** "Graphical programming", Page 555
CHF	Program a chamfer **Further information:** "Chamfer CHF", Page 164
L	Program a straight line segment **Further information:** "Straight line L", Page 162
CR	Program a circular arc with radius entry **Further information:** "Circular path CR", Page 172
RND	Program a rounding arc **Further information:** "Rounding RND", Page 166
CT	Program a circular arc with tangential connection to the preceding contour element **Further information:** "Circular path CT", Page 175
CC	Program a circle center or pole **Further information:** "Circle center point CC", Page 168
C	Program a circular arc with reference to the circle center **Further information:** "Circular path C ", Page 170
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools
CYCL DEF	In the **Insert NC function** window, open the **Fixed cycles** folder in order to select a cycle **Further information:** User's Manual for Machining Cycles
CYCL CALL	In the **Insert NC function** window, open the **Cycle call** folder in order to select a machining cycle **Further information:** User's Manual for Machining Cycles
LBL SET	Program a jump label **Further information:** "Defining a label with LBL SET", Page 222
LBL CALL	Program a subprogram or a program section repeat **Further information:** "Calling a label with CALL LBL", Page 223

Key	Meaning
STOP	Program an intentional stop **Further information:** "Programming the STOP function", Page 438
TOOL DEF	Pre-select a tool in the NC program **Further information:** "Tool pre-selection by TOOL DEF", Page 150
TOOL CALL	Call the tool data in the NC program **Further information:** "Tool call by TOOL CALL", Page 144
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)

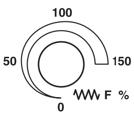
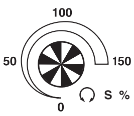
Keycaps for axis input and value input

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 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

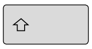
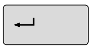
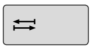
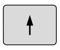
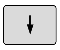
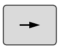
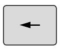
Key	Meaning
	Position the cursor
	
GOTO	■ Position the cursor by using the block number of an NC block ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one page up in an 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

Potentiometer	Function
	Increase or reduce the feed rate **Further information:** "Feed rate F", Page 149
	Increase or reduce the spindle speed **Further information:** "Spindle speed S", Page 148

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.
- Full-text search

  Search for functions in the workspace with the full-text search.

  Further information: "Adding and removing favorites", Page 77

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 59

  Further information: "Overview of workspaces", Page 62
- **Tools**

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

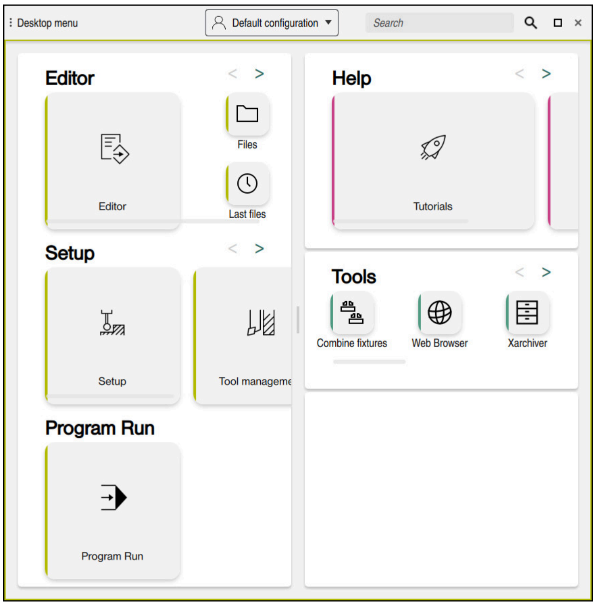
  Further information: User's Manual for Setup and Program Run
- **Help**

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

  Further information: "User's Manual as integrated product aid: TNCguide", Page 36
- **Favorites**

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

  Further information: "Adding and removing favorites", Page 77



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

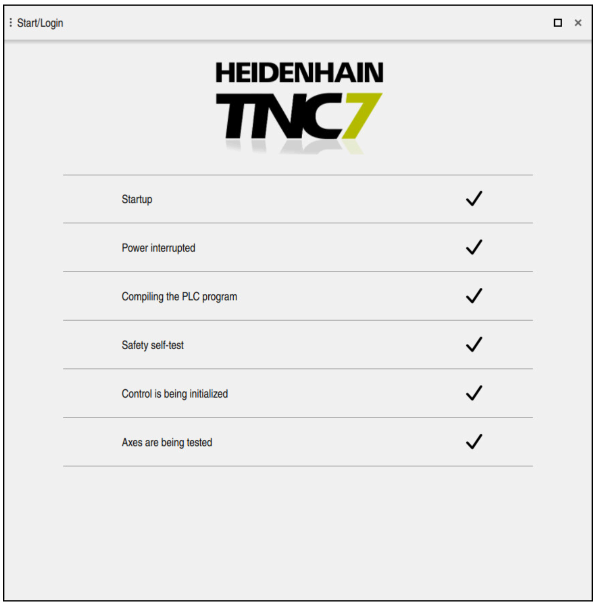
3.1 Chapter overview

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
- 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.

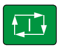
To switch the machine on:

- ▶ 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.

- ▶ 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: User's Manual for Setup and Program Run



- ▶ 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: User's Manual for Setup and Program Run

More detailed information

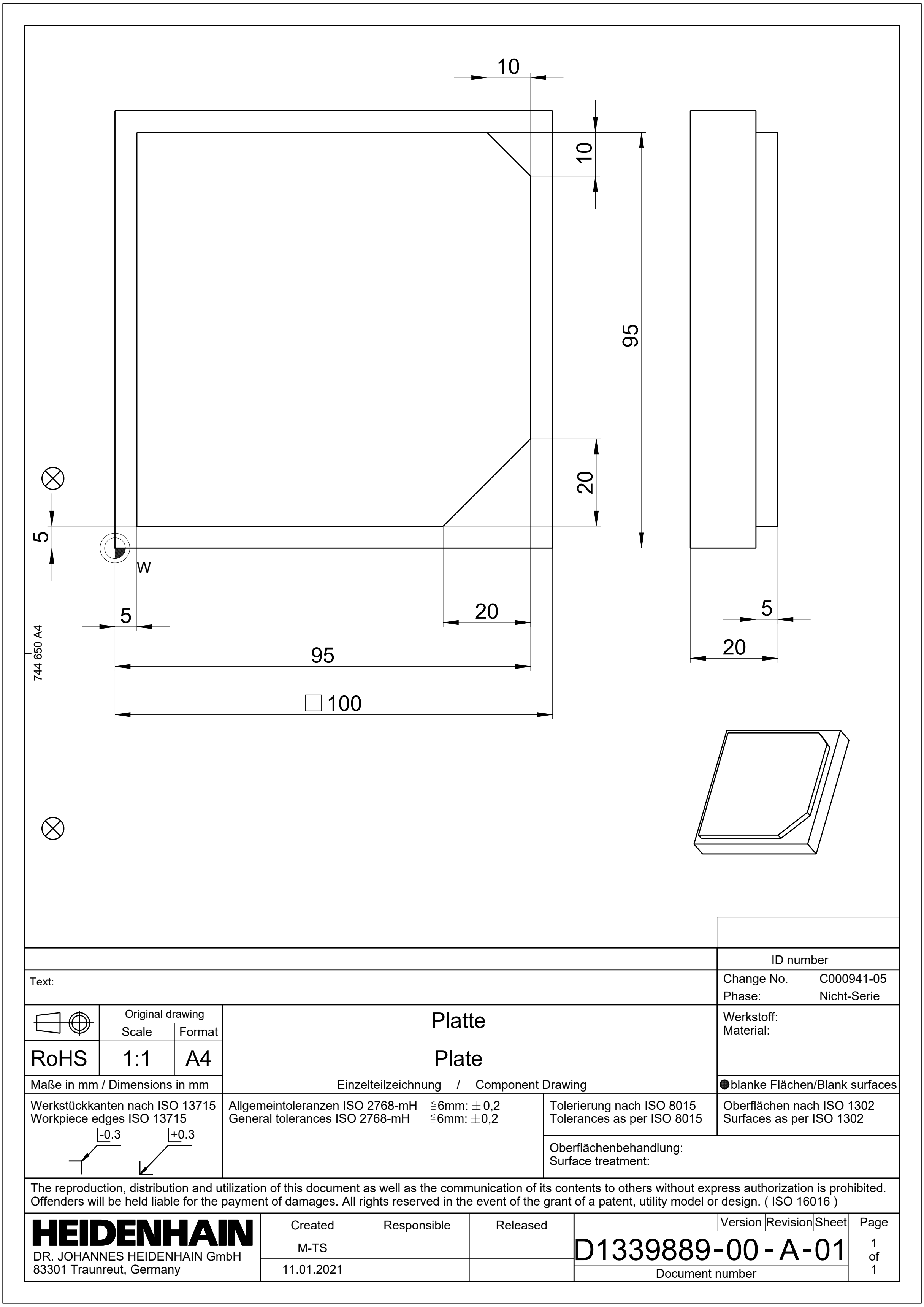
- Switching on and off

  Further information: User's Manual for Setup and Program Run
- Position encoders

  Further information: "Position encoders and reference marks", Page 103

3.3 Programming and simulating a workpiece

3.3.1 Example task 1339889



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 109

3.3.3 Configuring the control's user interface for programming

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

> The first steps describe the procedure when you are in the **Klartext 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

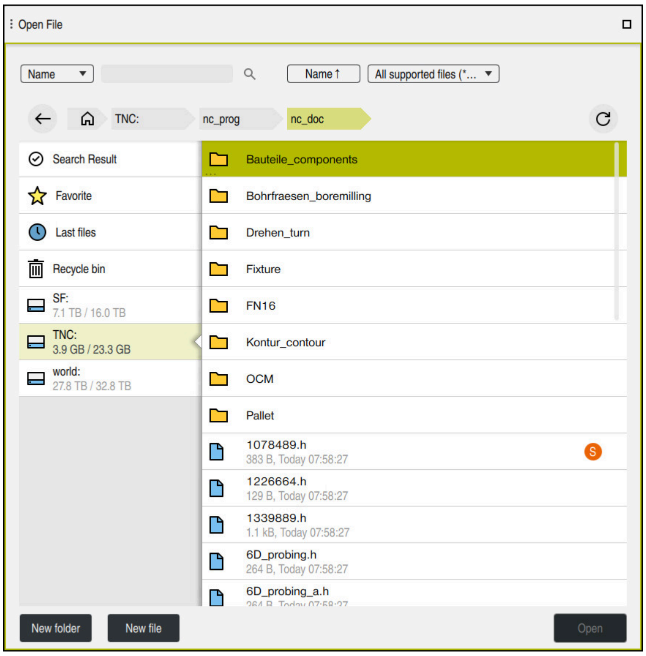
More detailed information

- Editing an NC program

  Further information: "Inserting and editing NC functions", Page 124
- The **Form** column

  Further information: "The Form column in the Program workspace", Page 121

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

New file

- Select **New file**
- Enter a file name (e.g., 1339899.h)

ENT

- Confirm with the **ENT** key

Open

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

More detailed information

- The **Open File** workspace

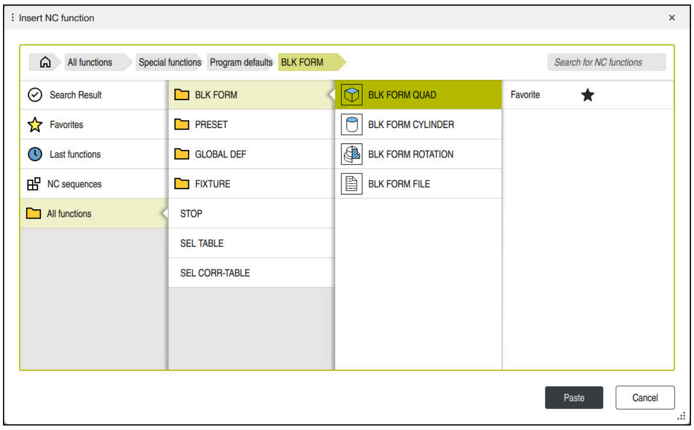
  Further information: User's Manual for Setup and Program Run
- The **Editor** operating mode

  Further information: "The Editor operating mode", Page 109

3.3.5 Defining the workpiece blank

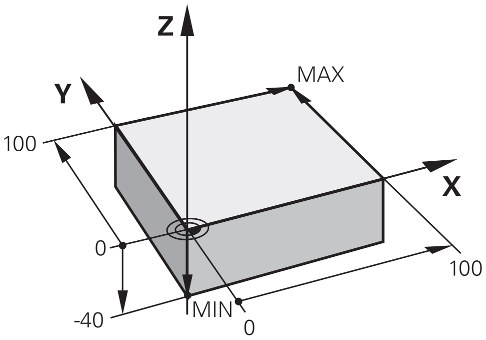
For the NC program you can define a workpiece blank that the control then uses for the simulation. When you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

> 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



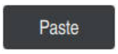
Cuboid workpiece blank with minimum point and maximum point

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

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

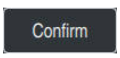
To define a cuboid workpiece blank:

- Select **BLK FORM QUAD**

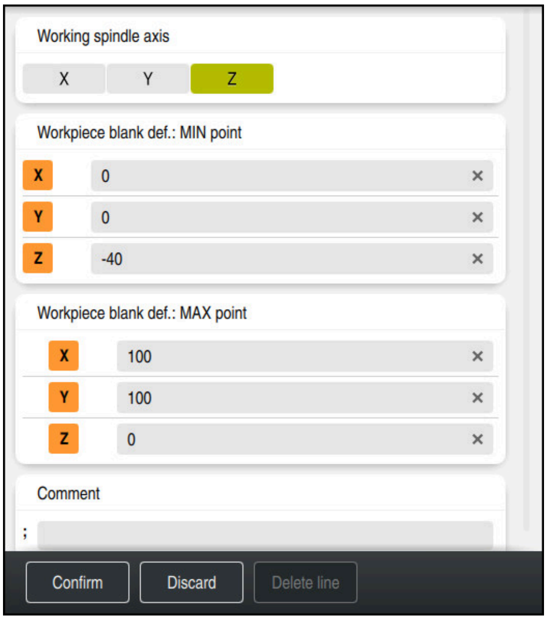


- Select **Paste**
- > The control inserts the NC block for definition of the workpiece blank.

- 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 132
- Reference points in the machine

  Further information: "Presets in the machine", Page 104

3.3.6 Structure of an NC program

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

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program

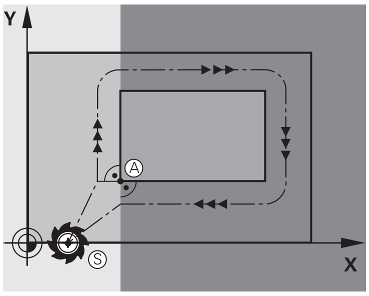
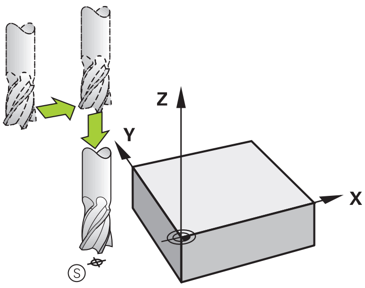
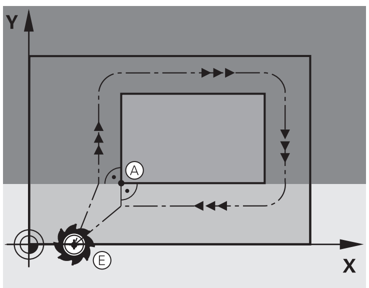
> The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

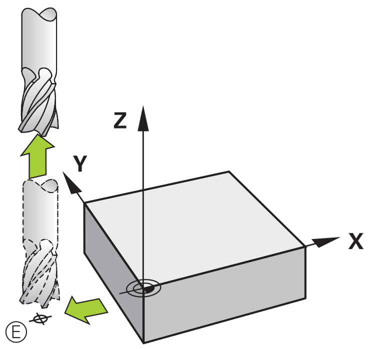
1. **BEGIN PGM** with selection of the unit of measure
2. Define the workpiece blank
3. Call the tool, with the tool axis and the technological data
4. Move the tool to a safe position, and switch the spindle on
5. Pre-position the tool in the working plane, near the first contour point
6. Pre-position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

3.3.7 Contour approach and departure

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

The following positions are necessary for contour approach and departure:

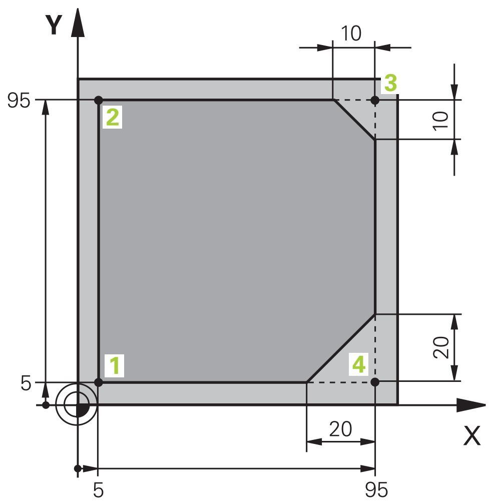
Help graphic	Position
	**Starting point** The following preconditions apply for the starting point: ■ No tool radius compensation ■ Approachable without danger of collision ■ Near to the first contour point The graphic shows the following information: If you define the starting point to be in the dark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

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

More detailed information

- Functions for approaching and departing from the contour
  Further information: "Fundamentals of approach and departure functions", Page 192

3.3.8 Programming a simple contour



Workpiece to be programmed

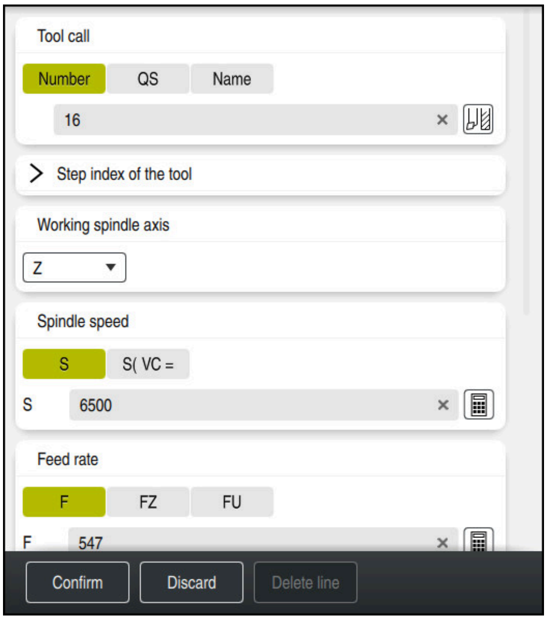
The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 85

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

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

Calling a tool



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**)

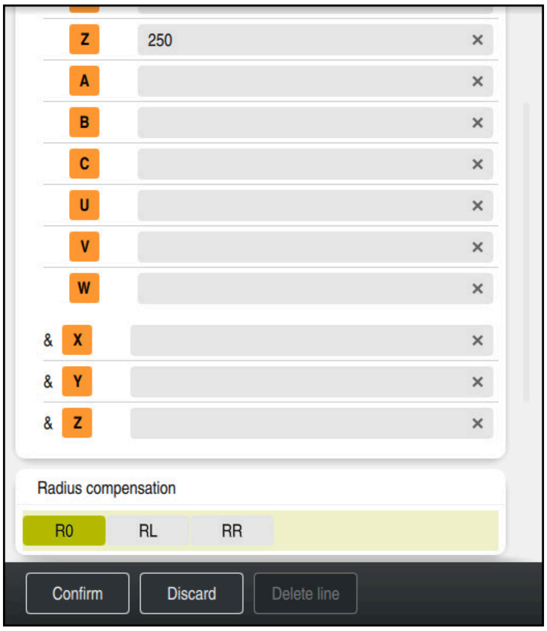
Confirm

- 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**

Z

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

Confirm

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

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

Pre-positioning in the working plane

To pre-position in the working plane:

- ▶ Select the path function **L**

X

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

Y

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

Confirm

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

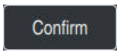
```
5  L  X-20 Y-20 FMAX
```

Pre-positioning in the tool axis

To pre-position in the tool axis:

- Select the path function **L**

- 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)

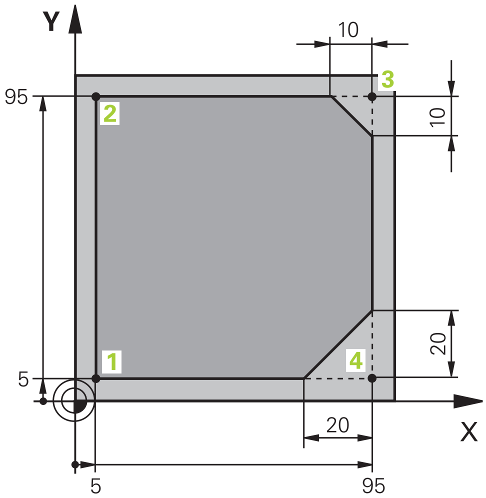


- Select **Confirm**

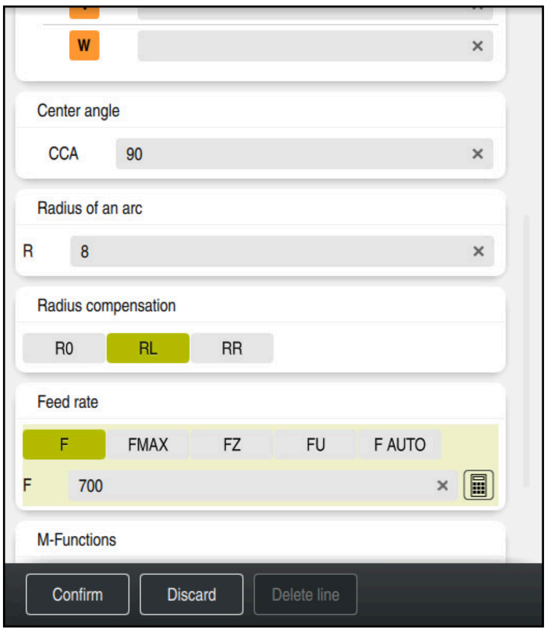
> The control concludes the NC block.

```
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

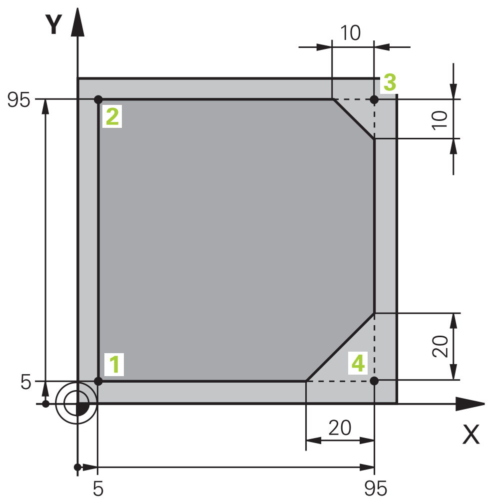
- ▶ Select **Paste**
- ▶ Enter the coordinates of the starting point **1** (e.g., **X 5 Y 5**)
- ▶ For the center angle **CCA**, enter the approach angle (e.g., **90**)
- ▶ Enter the radius of the circular arc (e.g., **8**)
- ▶ Select **RL**
- > The control applies tool radius compensation to the left.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

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

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

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

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

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

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

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

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

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

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

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

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

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

Departing from the contour

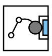
The **Form** column with the syntax elements of a departure function

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.

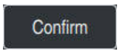
```
14  DEP CT  CCA90 R+8 F3000 M9
```

Moving the tool to a safe position 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)



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

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

More detailed information

- Tool call
  Further information: "Tool call by TOOL CALL", Page 144
- Line **L**
  Further information: "Straight line L", Page 162
- Designation of the axes and the working plane
  Further information: "Designation of the axes of milling machines", Page 102
- Functions for approaching and departing from the contour
  Further information: "Fundamentals of approach and departure functions", Page 192
- Chamfer **CHF**
  Further information: "Chamfer CHF", Page 164
- Miscellaneous functions
  Further information: "Overview of miscellaneous functions", Page 439

3.3.9 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.

> **i** 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.

> **i** 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.

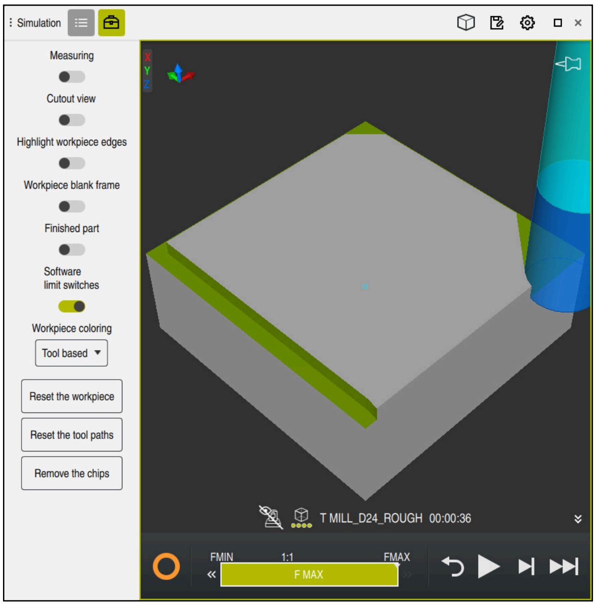
More detailed information

- The **Simulation** workspace
  Further information: "The Simulation Workspace", Page 629

3.3.10 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.

- 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 629

3.4 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

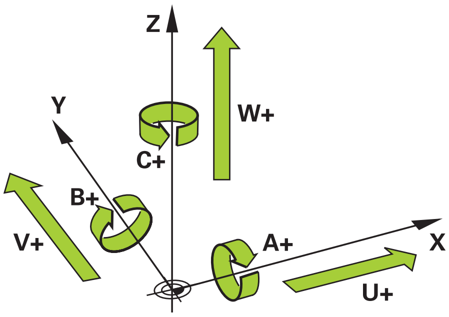
- Select **Shut down**
- > If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
- If necessary, save unsaved NC programs with **Save** or **Save as**
- > The control shuts down.
- > After completion of the shutdown process, the control displays the text **Now you can switch off.**
- Switch off the main power switch of the machine

4

NC and Programming Fundamentals

4.1 NC fundamentals

4.1.1 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.

4.1.2 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:

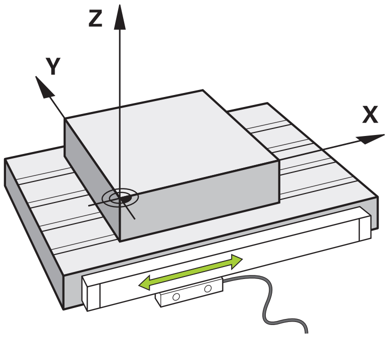
Main axis	Secondary axis	Tool axis	Working plane
X	**Y**	**Z**	**XY**, also **UV**, **XV**, **UY**
Y	**Z**	**X**	**YZ**, also **WU**, **ZU**, **WX**
Z	**X**	**Y**	**ZX**, also **VW**, **YW**, **VZ**

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.

4.1.3 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 236

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

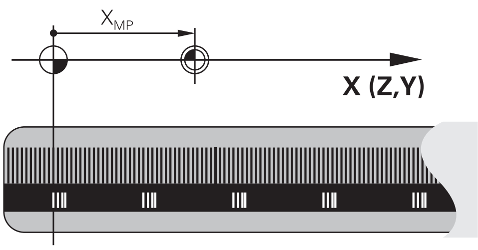
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

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: User's Manual for Setup and Program Run



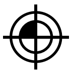
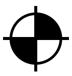
4.1.4 Presets in the machine

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 141

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 238 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 441
 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 442
	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 103 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** User's Manual for Setup and Program Run
	**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 243 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. 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.

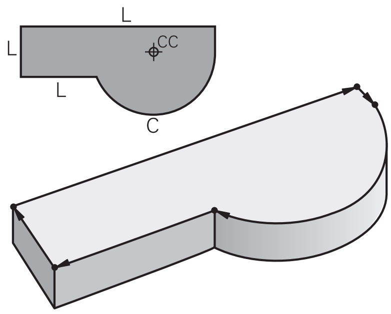
4.2 Programming possibilities

4.2.1 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 158



4.2.2 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 555

4.2.3 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 437

4.2.4 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 222

4.2.5 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 484

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 480

4.2.6 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 407

4.3 Programming fundamentals

4.3.1 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 84
- NC programs using CAD files

  Further information: "CAM-generated NC programs", Page 422
- Structure of an NC program for contour machining

  Further information: "Structure of an NC program", Page 87

Description of function

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

Further information: "The Program workspace", Page 110

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 component	Meaning
NC block	**4 TOOL CALL 5 Z S3200 F300** An NC block consists of the block number and the syntax of the NC function. An NC block can consist of multiple lines, such as with cycles. The control numbers the NC blocks in ascending sequence.
NC function	**TOOL CALL 5 Z S3200 F300** You use NC functions to define the behavior of the control. The block number is not a part of the NC functions.
Syntax initiator	**TOOL CALL** The syntax initiator clearly designates each NC function. Syntax initiators are used in the **Insert NC function** window. **Further information:** "Areas of the Insert NC function window", Page 122

Syntax component	Meaning
Syntax element	**TOOL CALL 5 Z S3200 F300** Syntax elements are all parts of the NC function, such as technology values **S3200** or coordinate information. NC functions also contain optional syntax elements. The control shows certain syntax elements in color in the **Program** workspace. **Further information:** "Appearance of the NC program", Page 112
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 106

4.3.2 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 602
- **Contour**

  Further information: "Graphical programming", Page 555
- **Program**

  Further information: "The Program workspace", Page 110
- **Simulation**

  Further information: "The Simulation Workspace", Page 629
- **Simulation status**

  Further information: User's Manual for Setup and Program Run
- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 604

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 654

Further information: "The Form workspace for pallets", Page 662

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 654

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 555
	The control uses this icon to show that a pallet table is open. **Further information:** "Pallet Machining and Job Lists", Page 653
	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 124
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Inserting and editing NC functions", Page 124
GOTO block number	The control selects the block number that you defined. **Further information:** "GOTO function", Page 607
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 484
/ 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 609
; 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 608
Edit	The control opens the context menu. **Further information:** "Context menu", Page 618
Select in Program Run	The control opens the file in the **Program Run** operating mode. **Further information:** User's Manual for Setup and Program Run
Start the simulation	The control opens the **Simulation** workspace and starts graphic simulation. **Further information:** "The Simulation Workspace", Page 629

4.3.3 The Program workspace

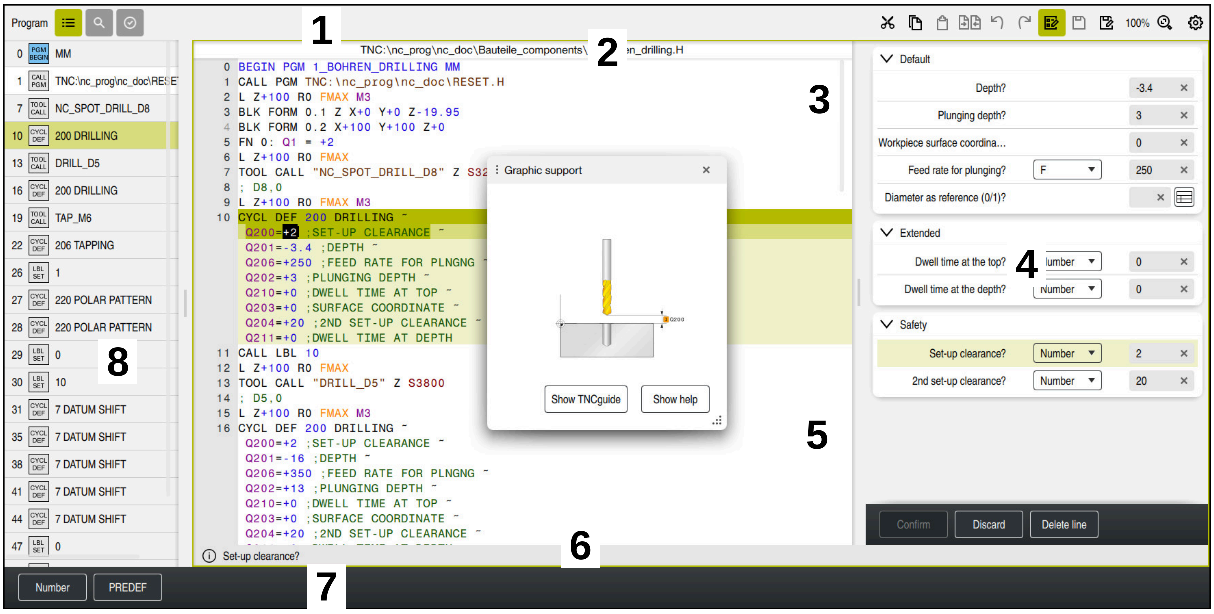
Application

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 112

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.

3 Contents of the NC program

   Further information: "Appearance of the NC program", Page 112

4 **Form** column

   Further information: "The Form column in the Program workspace", Page 121

5 Help graphic of the syntax element being edited

   Further information: "Help graphic", Page 113

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 610

   Further information: "The Search column in the Program workspace", Page 613

   Further information: User's Manual for Setup and Program Run

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 74

Icon or shortcut	Function
	Open and close the **Structure** column **Further information:** "The Structure column in the Program workspace", Page 610
CTRL + F	Open and close the **Search** column **Further information:** "The Search column in the Program workspace", Page 613
	Open and close the **Tool check** column **Further information:** User's Manual for Setup and Program Run
	Activate and end comparison functions **Further information:** "Program comparison", Page 616
	Open and close the **Form** column **Further information:** "The Form column in the Program workspace", Page 121
100%	Font size of the NC program 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 113

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 113

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 36
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 602

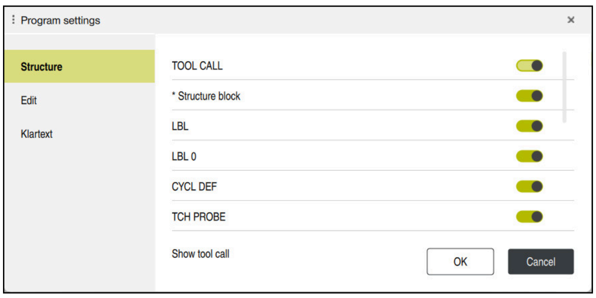
Settings in the Program workspace

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.

Further information: "The Structure column in the Program workspace", Page 610

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 125
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 126
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 355

Setting	Meaning
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. When you select the **All Files (*.*)** filter, the control displays the file in the **Open File** workspace. 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 602
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 230
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).

Setting	Meaning
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 608
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 144 ■ Preselect a tool with **TOOL DEF** **Further information:** "Tool pre-selection by TOOL DEF", Page 150 **Further information:** User's Manual for Setup and Program Run
Skip linear superim-posed 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 170 ■ Circular contour **CR** **Further information:** "Circular path CR", Page 172 ■ Circular contour **CT** **Further information:** "Circular path CT", Page 175 **Further information:** "Linear superimpositioning of a circular path", Page 177

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 694
- **Tool correction**

  Further information: "Compensation table *.tco", Page 704
- **Workpiece correction**

  Further information: "Compensation table *.wco", Page 706

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 502

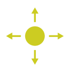
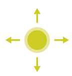
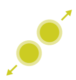
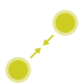
Using the Program workspace

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

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

Touch operation

You use gestures to perform the following functions:

Symbol	Gesture	Meaning
	Tap	■ Select an NC block ■ Select a syntax element while editing
	Double tap	Edit an NC block
	Long press	Open the context menu If you are working with a mouse, click with the right mouse key. **Further information:** "Context menu", Page 618
	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 621
	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 120
▶ ◀	■ 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 607 ■ 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 126
Insert NC function	Open the **Insert NC function** window **Further information:** "Areas of the Insert NC function window", Page 122
Edit	Open the context menu **Further information:** "Context menu", Page 618

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.

> 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 662
- Editing an NC function in the **Form** column

  Further information: "Editing NC functions", Page 126

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.

4.3.4 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

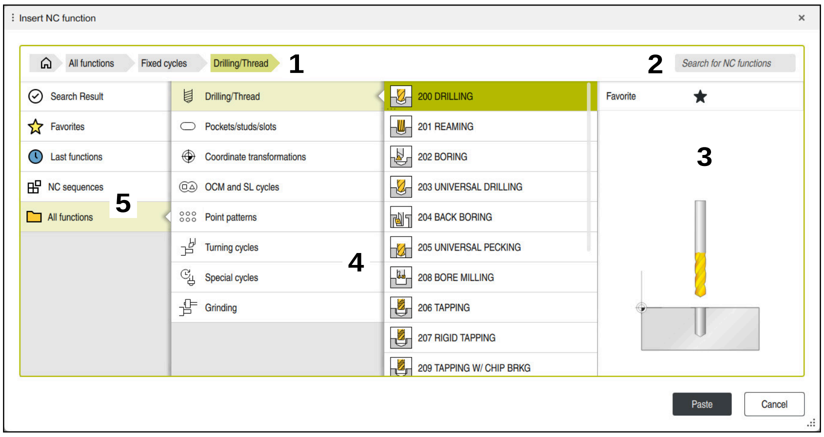
- Creating NC sequences
  Further information: "NC sequences for reuse", Page 230
- Inserting and editing NC functions
  Further information: "Inserting and editing NC functions", Page 124

Description of function

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

> 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 352
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**.

  > 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 74
- **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 230
- **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 69

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 230

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.

4.3.5 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 118
- **Insert NC function** window

  Further information: "The Insert NC function window", Page 122

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 604
- 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 122
- 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 125

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 126

Inserting NC functions

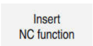
Inserting an NC function directly with keys or buttons

To insert frequently needed NC functions:

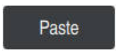
- ▶ Select **L**
- > The control creates a new NC block and starts the dialog.
- ▶ Follow the instructions in the dialog

Inserting an NC function by selecting it

To insert a new NC function:



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



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

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 113
- ▶ 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.

Yes

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

> 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 120
- 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 74
- 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: User's Manual for Setup and Program Run
- 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 118
- 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: User's Manual for Machining Cycles

5

Technology-Specific NC Programming

5.1 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: User's Manual for Setup and Program Run

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.

6

Workpiece Blank

6.1 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.

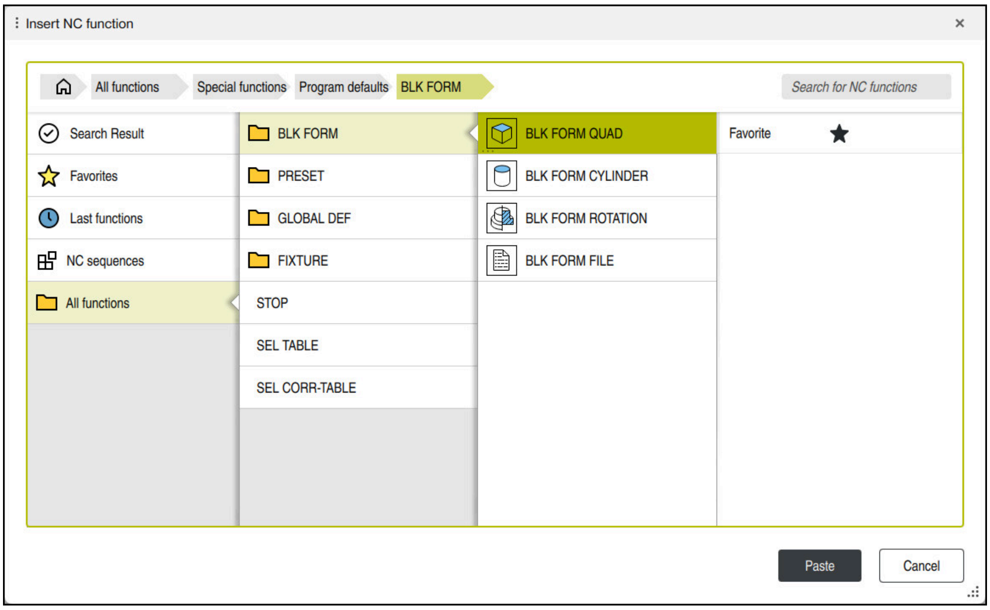
Related topics

- Representation of the workpiece blank in the **Simulation** workspace
  Further information: "The Simulation Workspace", Page 629

Description of function

You define the blank relative to the workpiece preset.

Further information: "Presets in the machine", Page 104



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 84

The control offers the following workpiece blank definitions:

Icon	Meaning	Further information
	**BLK FORM QUAD** Cuboid workpiece blank	Page 134
	**BLK FORM CYLINDER** Cylindrical workpiece blank	Page 135
	**BLK FORM ROTATION** Rotationally symmetric blank with a definable contour	Page 136
	**BLK FORM FILE** STL file as workpiece blank and finished part	Page 137

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 388
- 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.

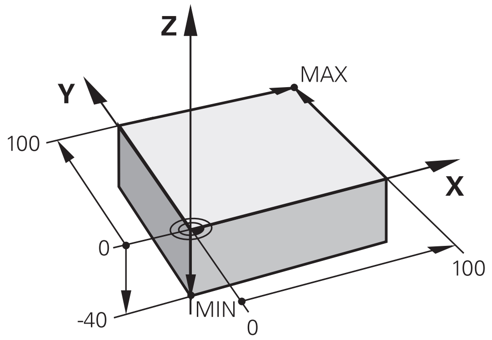
  Further information: "Exporting a simulated workpiece as STL file", Page 641

6.1.1 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 104

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:

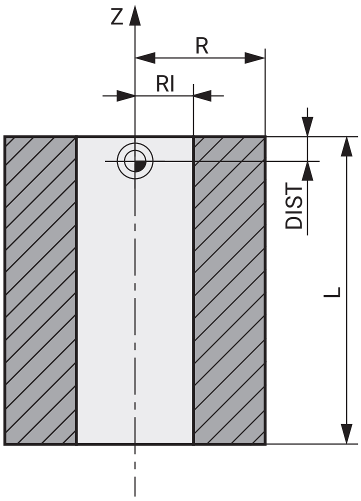
Syntax element	Meaning
BLK FORM	Start of syntax for cuboid workpiece blank
0.1	Designation of the first NC block
Z	Tool axis Other possibilities might be available, depending on the machine.
X Y Z	Coordinate definition of the MIN point
0.2	Designation of the second NC block
X Y Z	Coordinate definition of the MAX point

6.1.2 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:

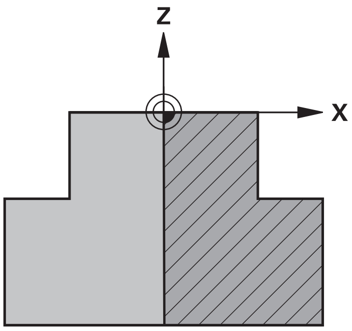
Syntax element	Meaning
BLK FORM CYLINDER	Syntax initiator for cylindrical workpiece blank
Z	Rotary axis Other possibilities might be available, depending on the machine.
R or **D**	Radius or diameter of the cylinder
L	Total height of the cylinder
DIST	Oversize of the cylinder relative to the workpiece preset Optional syntax element
RI or **DI**	Inside radius diameter of the core hole Optional syntax element

6.1.3 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: User's Manual for Setup and Program Run

6.1.4 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 641
- 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 647
- 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.

7

Tools

7.1 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 141
- Further tool data are needed to completely define the tool. Take these tool data from the manufacturer's tool catalog, for example.

  Further information: User's Manual for Setup and Program Run
- Save all collected tool data of this tool in the tool management.

  Further information: User's Manual for Setup and Program Run
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: User's Manual for Setup and Program Run
- After finishing tool definition, program a tool call within an NC program.

  Further information: "Tool call by TOOL CALL", Page 144
- 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 150
- 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: User's Manual for Setup and Program Run
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: "Tool radius compensation", Page 326

7.2 Presets on the tool

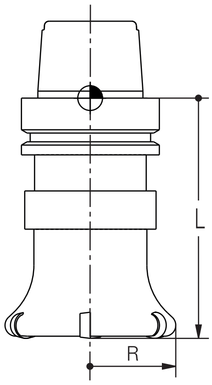
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

  Further information: "Presets in the machine", Page 104

7.2.1 Tool carrier reference point

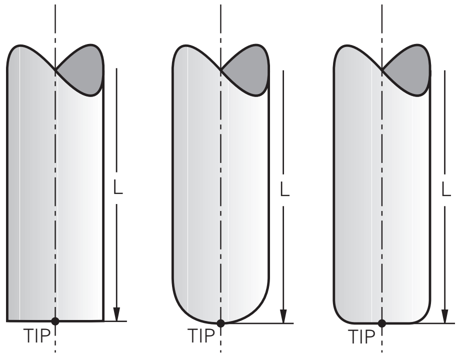


The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

Starting from the tool carrier reference point, define the tool dimensions in the tool management (e.g., length **L** and radius **R**).

Further information: User's Manual for Setup and Program Run

7.2.2 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 249

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:

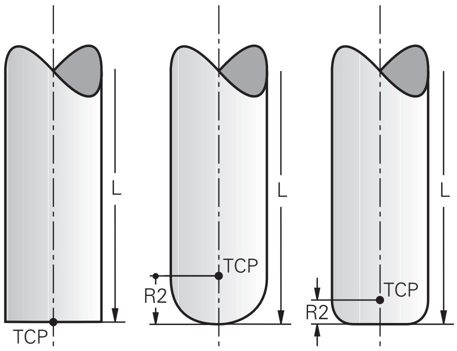
- **L**
- **DL**

Further information: User's Manual for Setup and Program Run

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 143

7.2.3 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.

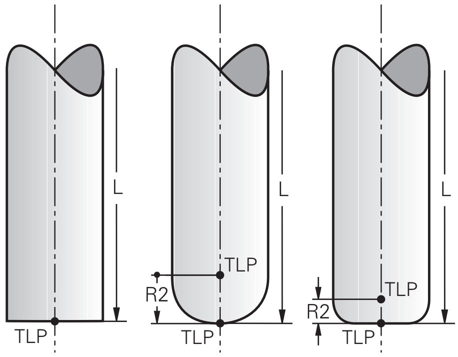
Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: User's Manual for Setup and Program Run

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 143

7.2.4 Tool location point (TLP, tool location point)

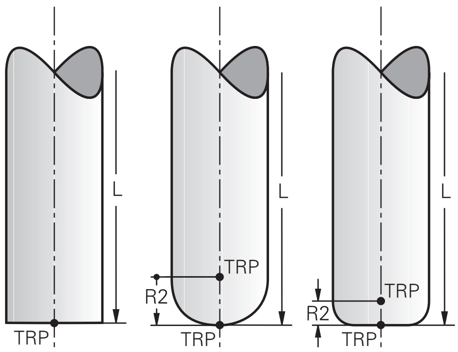


The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

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 315

7.2.5 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.

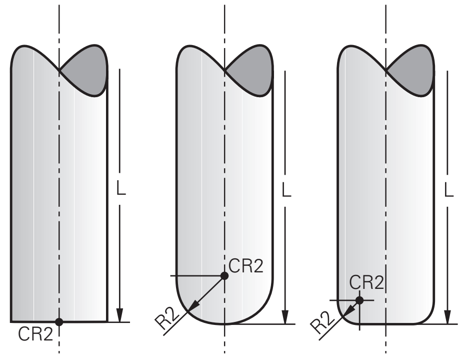
When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: "Rotary axis positioning", Page 303

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.

Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315

7.2.6 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 333

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 143

7.3 Tool call

7.3.1 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 473
- Tool table **tool.t**

  Further information: User's Manual for Setup and Program Run
- Pocket table **tool_p.tch**

  Further information: User's Manual for Setup and Program Run

Requirement

- Tool defined

  To call a tool, the tool must be defined in the tool management.

  Further information: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

> 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 439

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.
	Open the **Cutting data calculator** **Further information:** "Cutting data calculator", Page 625

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 147
.1	Step index of the tool Optional syntax element **Further information:** User's Manual for Setup and Program Run
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 147
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 148
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 149
DL	Delta value of tool length Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 324

Syntax element	Meaning
DR	Delta value of the tool radius Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 324
DR2	Delta value of the tool radius 2 Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 324

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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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

> 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 473

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.

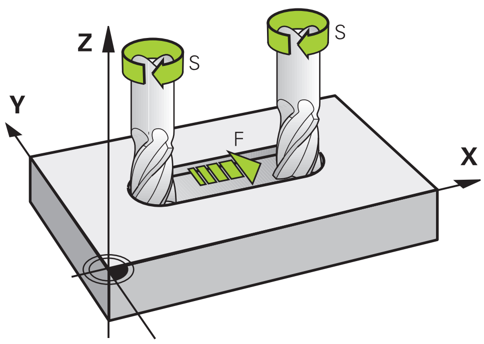
  Further information: "Tool pre-selection by TOOL DEF", Page 150
- The machine manufacturer uses the optional machine parameter **prog-ToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

  Further information: "Tool compensation for tool length and tool radius", Page 324

7.3.2 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 144
- **S** button in the **Manual operation** application

  Further information: User's Manual for Setup and Program Run

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 72

Status displays

The control displays the current spindle speed in the following workspaces:

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

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 144
- Positioning block
  Further information: "Path Functions", Page 153
- **F** button in the **Manual operation** application
  Further information: User's Manual for Setup and Program Run

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:** User's Manual for Setup and Program Run

- 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 149

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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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 72

Status displays

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

- The **Positions** workspace
- 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.

- 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: User's Manual for Setup and Program Run

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.

7.3.3 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 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:** User's Manual for Setup and Program Run

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

8

Path Functions

8.1 Fundamentals of coordinate definitions

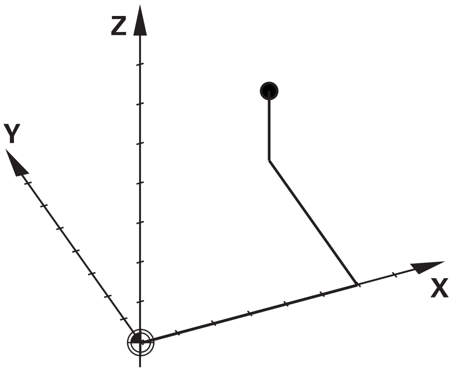
You program a workpiece by defining the path contours and the target coordinates.

Depending on the dimensioning used in the technical drawing, you use Cartesian or polar coordinates with absolute or incremental values.

8.1.1 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
```

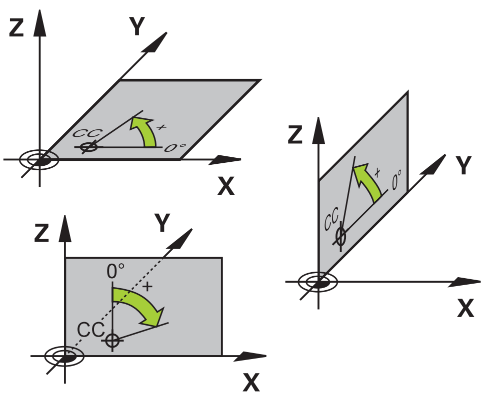
The programmed coordinates are modally effective. As long as the value of an axis remains the same, you do not need to program the value for further path contours.

8.1.2 Polar coordinates

Application

You define polar coordinates in one of the three planes of a Cartesian coordinate system.

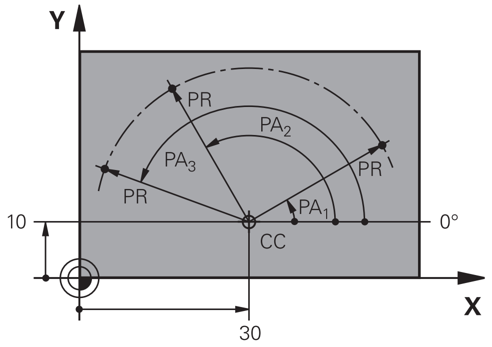
Polar coordinates are relative to a previously defined pole. From this pole you define a point by its distance to the pole and the angle to the angle reference axis.



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.

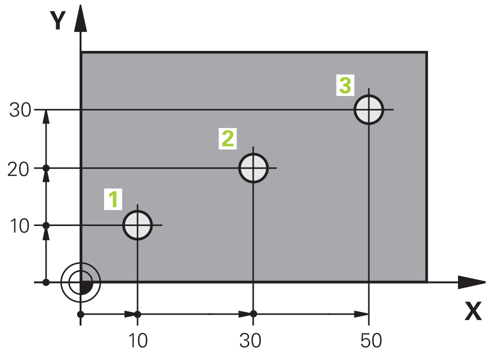
8.1.3 Absolute input

Application

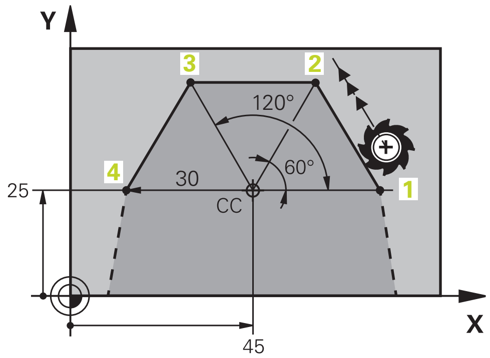
Absolute input always references an origin. For Cartesian coordinates, the origin is the datum, and for polar coordinates the origin is the pole and the angle reference axis.

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

8.1.4 Incremental entries

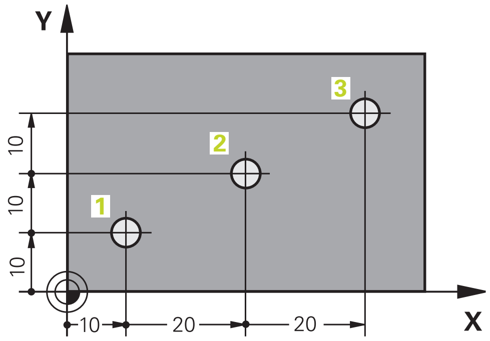
Application

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**.

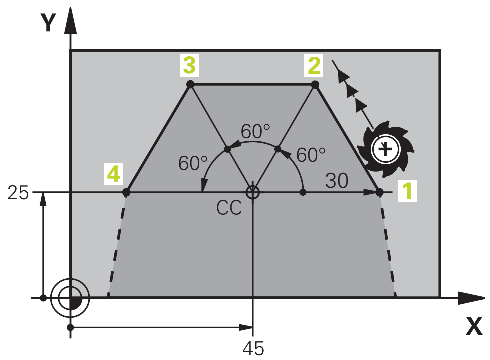
Description of function

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

8.2 Fundamentals of path functions

Application

When creating an NC program, you can use the path functions to program the individual contour elements. To do so, use coordinates to define the end points of the contour elements.

The control then uses the coordinate entries, the tool data, and the radius compensation to calculate the traverse path. The control simultaneously positions all machine axes that you programmed in the NC block of a path function.

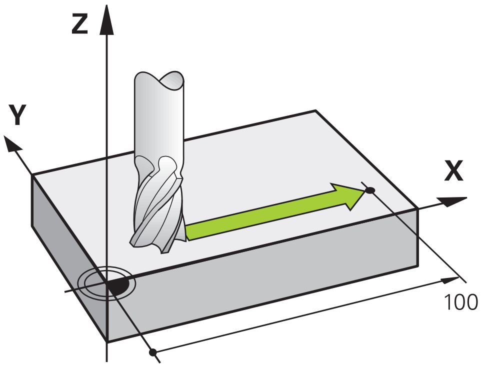
Description of function

Inserting a path function

The gray path function keys initiate the dialog. The control inserts the NC block in the NC program and prompts you for each piece of necessary information.

> Depending on the design of the machine tool, either the tool moves or the machine table moves. When programming a path function, you always assume that the tool is in motion.

Motion in one axis



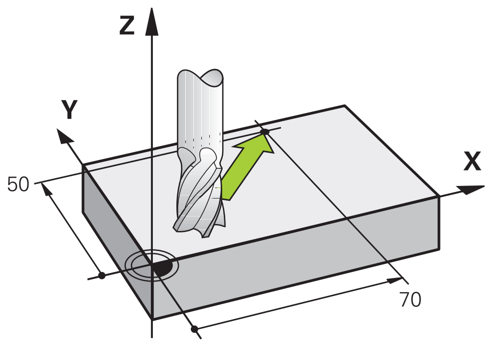
If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Example

```
L  X+100
```

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

Motion in two axes



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

Example

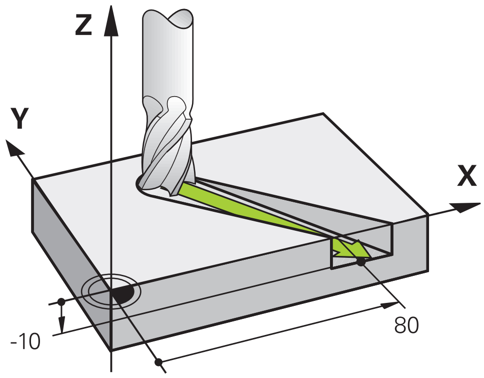
```
L X+70  Y+50
```

The tool retains the Z coordinate and moves in the XY plane to the position **X+70 Y+50**.

You define the working plane by entering the tool axis when calling the tool with **TOOL CALL**.

Further information: "Designation of the axes of milling machines", Page 102

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
```

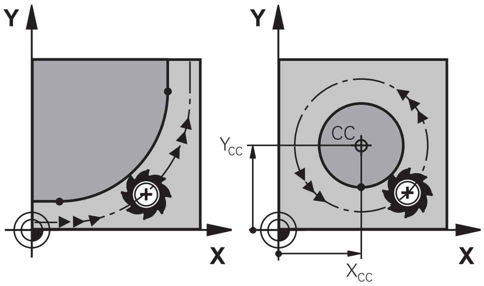
Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

Example

```
L X+80 Y+0 Z-10 A+15 B+0 C-45
```

> 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.

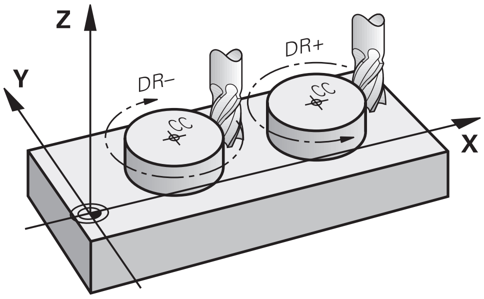
Circles and arcs



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 326

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

8.3 Path functions with Cartesian coordinates

8.3.1 Overview of path functions

Key	Function	Further information
L	Straight line **L** (line)	Page 162
CHF	Chamfer **CHF** (chamfer) Chamfer between two straight lines	Page 164
RND	Rounding **RND** (rounding of corner) Circular arc with tangential connection to the preceding and subsequent contour elements	Page 166
CC	Circle center point **CC** (circle center)	Page 168
C	Circular path **C** (circle) Circular path around a circle center **CC** to an end point	Page 170
CR	Circular path **CR** (circle by radius) Circular path with a specified radius	Page 172
CT	Circular path **CT** (circle tangential) Circular path with tangential connection to the preceding contour element	Page 175

8.3.2 Straight line L

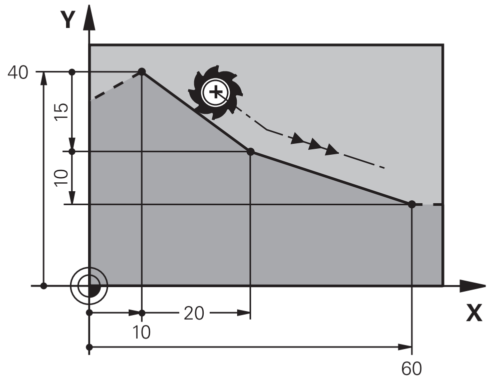
Application

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 182

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 413 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121
- 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.

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
```

8.3.3 Chamfer CHF

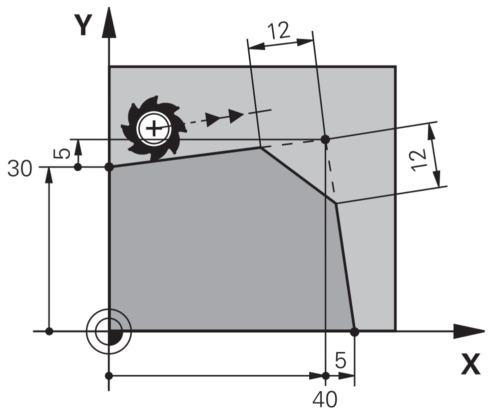
Application

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 149 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
```

8.3.4 Rounding RND

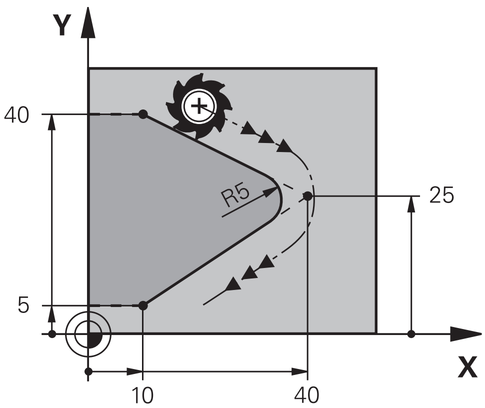
Application

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 149 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
```

8.3.5 Circle center point CC

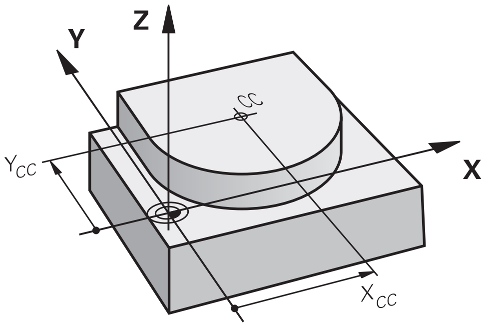
Application

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 181

Description of function



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

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

> The control simultaneously uses the **CC** function as the pole for polar coordinates.
> **Further information:** "Polar coordinate datum at pole CC", Page 181

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
```

8.3.6 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

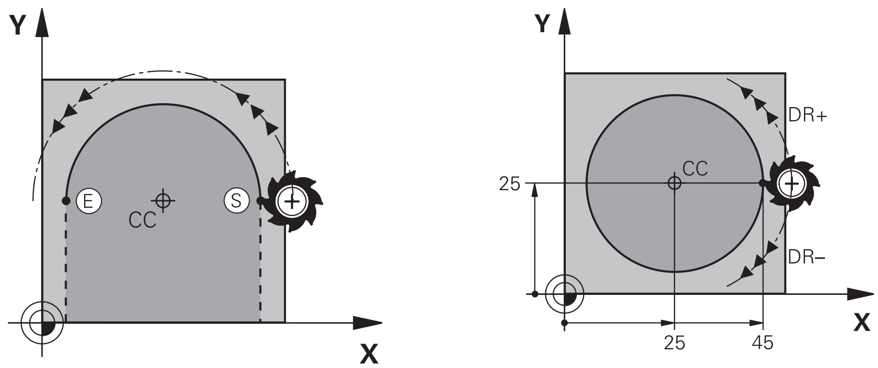
  Further information: "Circular path CP around pole CC", Page 185

Requirement

- Circle center point **CC** is defined

  Further information: "Circle center point CC", Page 168

Description of function



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

If you want to program a full circle, then define the same coordinates for the starting and end point. These points must lie on the circular path.

> In the machine parameter **circleDeviation** (no. 200901) you can define the permissible deviation of the circle radius. The maximum permissible deviation is 0.016 mm.

With the direction of rotation you define whether the control moves along the circular path in a clockwise or counterclockwise direction.

Definition of the direction of rotation:

- Clockwise: direction of rotation **DR-** (with radius compensation **RL**)
- Counterclockwise: direction of rotation **DR+** (with radius compensation **RL**)

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 177 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

Example

```
5 CC  X+25 Y+25
6 L  X+45 Y+25  RR F200  M3
7 C  X+45 Y+25  DR+
```

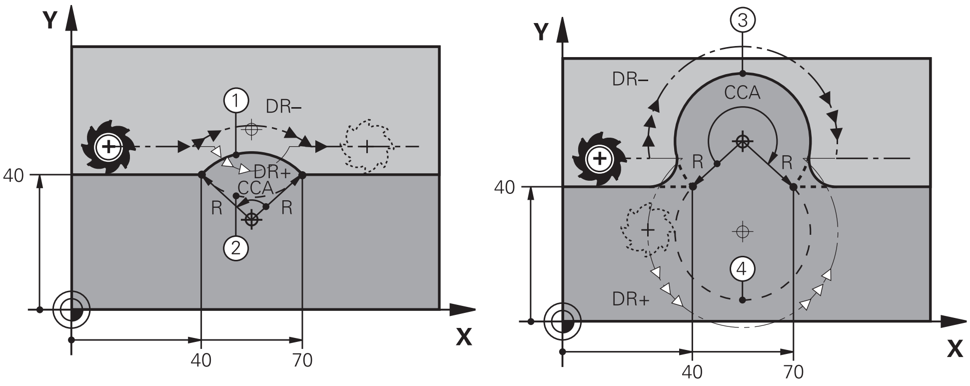
8.3.7 Circular path CR

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

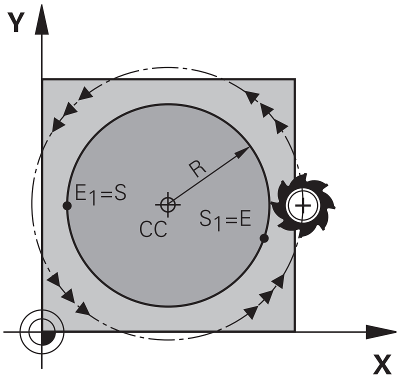
11 CR X+70 Y+40 R+20 DR+	; Circular path 2

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 177 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 Fixed or variable number Optional syntax element

Note

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

8.3.8 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

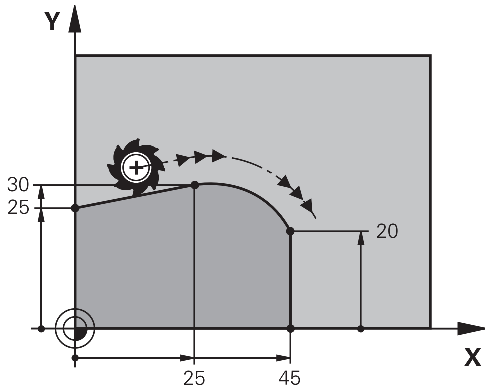
  Further information: "Circular path CTP", Page 187

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 177 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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
```

8.3.9 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 189

Description of function

You can linearly superimpose the following circular paths:

- Circular contour **C**

  Further information: "Circular path C ", Page 170
- Circular contour **CR**

  Further information: "Circular path CR", Page 172
- Circular contour **CT**

  Further information: "Circular path CT", Page 175

> 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 113
- 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 162

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.

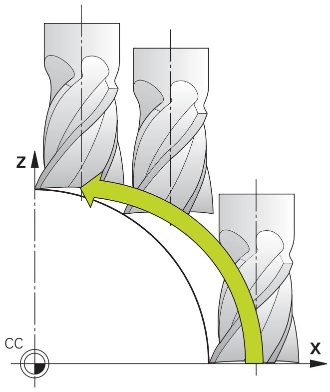
Further information: "Subprograms and program section repeats with the label LBL", Page 222

8.3.10 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 102

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

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

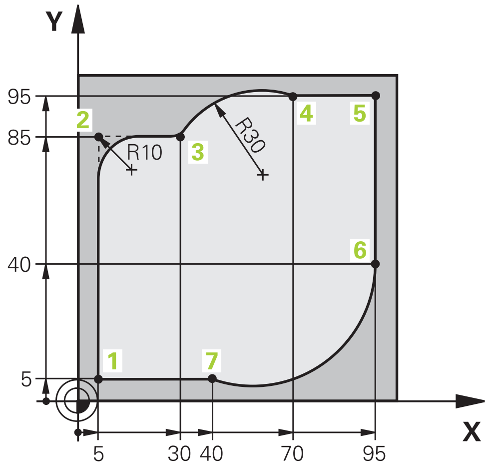
> If you want to use the function **C** for circular paths in another plane, you must first define the circle center point **CC** by entering one of the axes of the working plane and the tool axis.

Spatial arcs are created when these circular paths rotate. When machining spatial arcs, the control moves in three axes.

Example

```
3  TOOL CALL 1 Z S4000
4 ...
5 L  X+45 Y+25 Z+25 RR F200  M3
6 CC  X+25 Z+25
7 C  X+45 Z+25  DR+
```

8.3.11 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	

8.4 Path functions with polar coordinates

8.4.1 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 182
C + P	Circular path **CP** (circle polar) Circular path around circle center point or pole **CC** to arc end point	Page 185
CT + P	Circular path **CTP** (circle tangential polar) Circular path with tangential connection to the preceding contour element	Page 187
C + P	Helix with circular path **CP** (circle polar) Combination of a circular and a linear motion	Page 189

8.4.2 Polar coordinate datum at pole CC

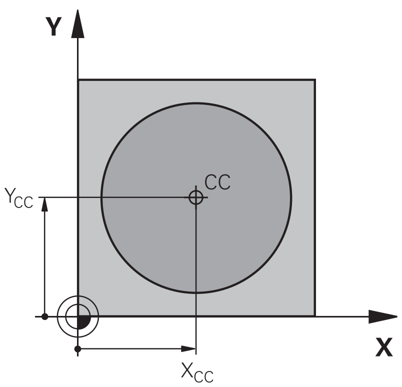
Application

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 168

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
```

8.4.3 Straight line LP

Application

With the straight line function **LP** you program a straight traverse motion in any direction using polar coordinates.

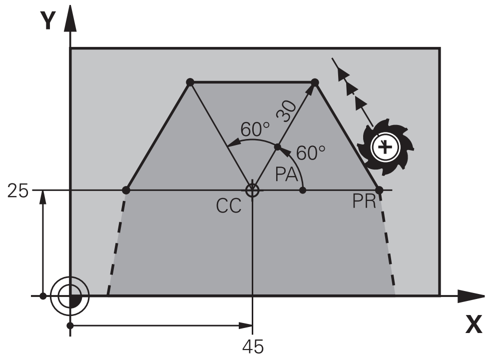
Related topics

- Programming a straight line with Cartesian coordinates
  Further information: "Straight line L", Page 162

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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
```

8.4.4 Circular path CP around pole CC

Application

You use the circular path function **CP** to program a circular path around the defined pole.

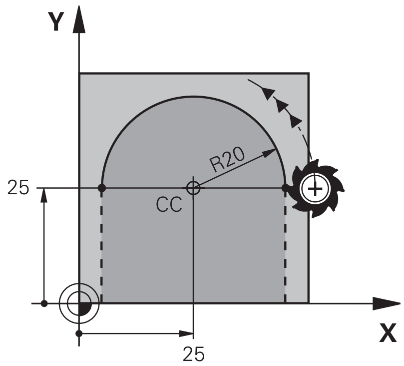
Related topics

- Programming a circular path with Cartesian coordinates
  Further information: "Circular path C ", Page 170

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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 189 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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+
```

8.4.5 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 175

Requirements

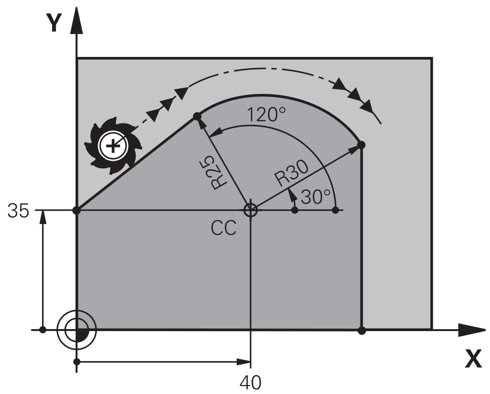
- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 181
- 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 189 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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
```

8.4.6 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

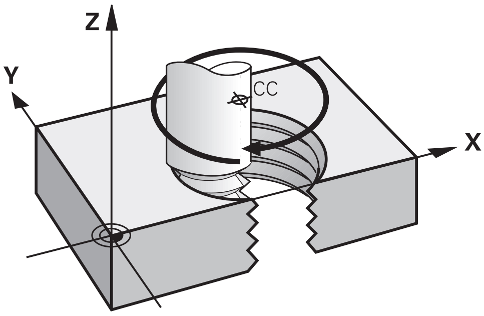
  Further information: "Linear superimpositioning of a circular path", Page 177

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 185

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

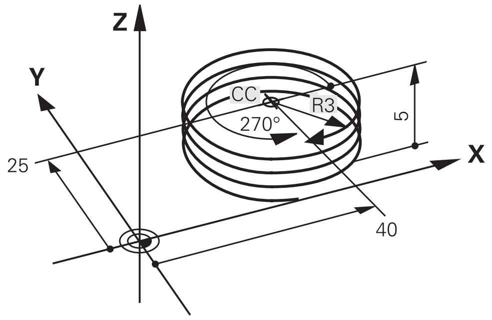
Dependencies of different thread shapes

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.

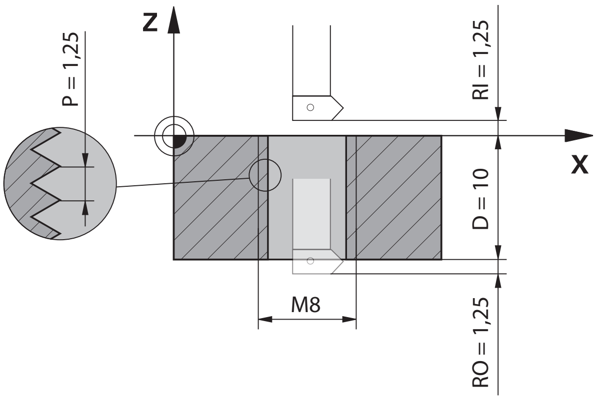
To program a helix:

- 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 190

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.

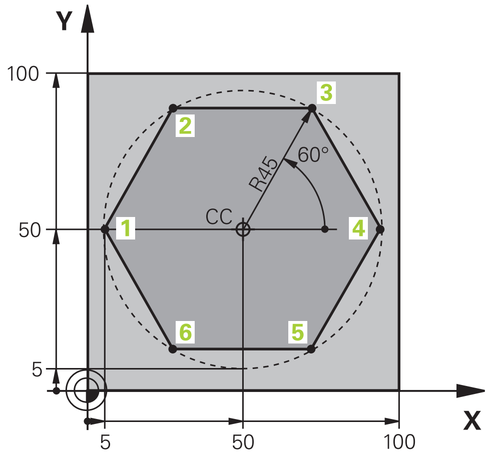
11 L Z+1,25 R0 FMAX	; Pre-position in the tool axis
12 L X+4 Y+0 RR F500	; Pre-position in the plane
13 CC X+0 Y+0	; Activate the pole
14 CP IPA-3600 IZ-12.5 DR-	; Cut the thread

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

Further information: "Subprograms and program section repeats with the label LBL", Page 222

Further information: "Example", Page 178

8.4.7 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	

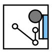
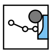
8.5 Fundamentals of approach and departure functions

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

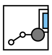
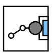
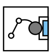
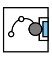
Because the approach and departure functions encompass multiple path functions, you get shorter NC programs. The defined syntax elements **APPR** and **DEP** make it easier for you to find contours in the NC program.

8.5.1 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 195
	**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 198
	**APPR CT** or **APPR PCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection	Page 200
	**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 202

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 204
	**DEP LN** Depart contour on a straight line perpendicular to the last contour point	Page 205
	**DEP CT** Depart contour on a circular path with a tangential connection	Page 206
	**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 206

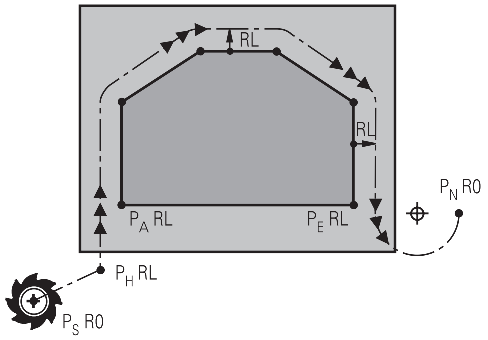
> You can switch between entry of Cartesian and polar coordinates in the form or by pressing the **P** key.
>
> **Further information:** "Fundamentals of coordinate definitions", Page 154

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 superimpositioning of a circular path", Page 189

8.5.2 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**.

  > 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

8.6 Approach and departure functions with Cartesian coordinates

8.6.1 Approach function APPR LT

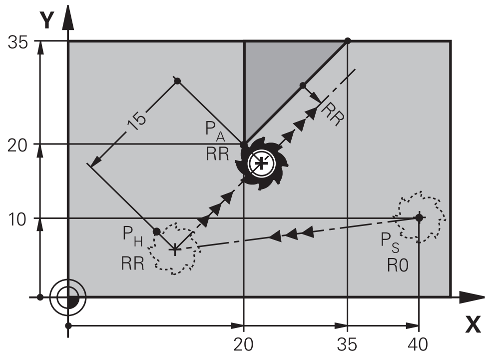
Application

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 209

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.6.2 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.

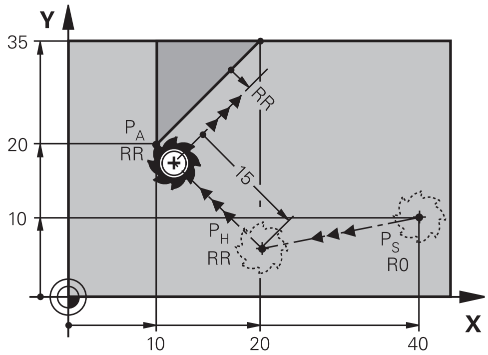
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PLN** with polar coordinates

  Further information: "Approach function APPR PLN", Page 211

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.6.3 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.

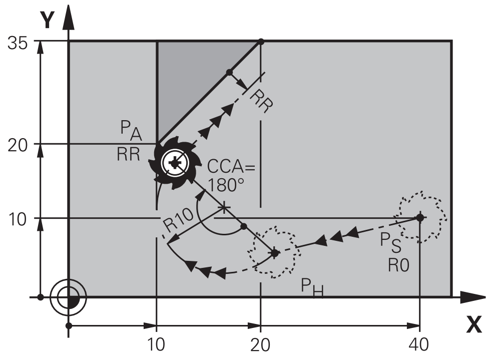
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PCT** with polar coordinates

  Further information: "Approach function APPR PCT", Page 213

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

> 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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.6.4 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.

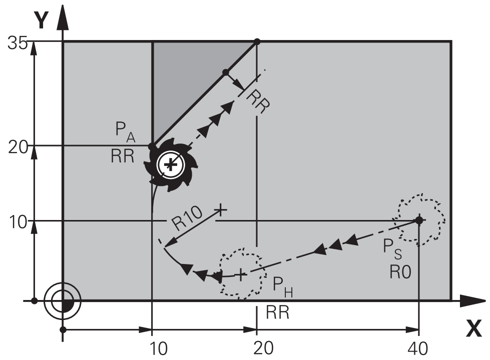
Coordinates of the first contour point are programmed with Cartesian coordinates.

Related topics

- **APPR PLCT** with polar coordinates

  Further information: "Approach function APPR PLCT", Page 216

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

Example APPR LCT

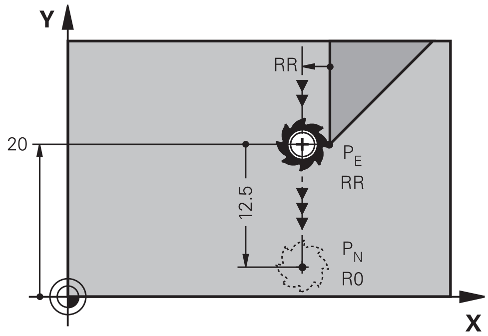
11 L X+40 Y+10 R0 F300 M3	; Approach P_S with **R0**
12 APPR LCT X+10 Y+20 Z-10 R10 RR F100	; Approach P_A with **RR**; distance P_H to P_A: **R10**
13 L X+20 Y+35	; Complete the first contour element

8.6.5 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 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 Fixed or variable number Optional syntax element

Example DEP LT

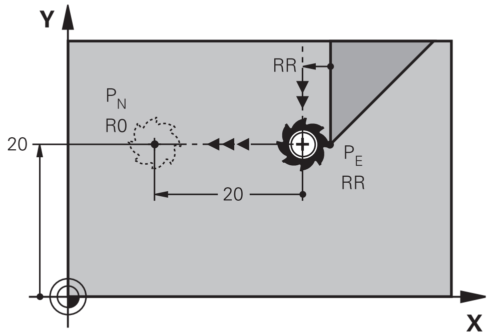
11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LT LEN12.5 F100	; Approach P_N; distance P_E to P_N: **LEN12.5**

8.6.6 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 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 Fixed or variable number Optional syntax element

Example DEP LN

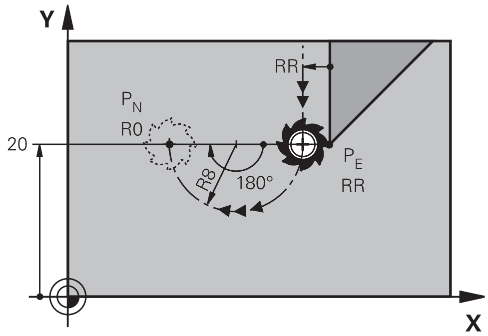
11 L Y+20 RR F100	; Approach the last contour element P_E with **RR**
12 DEP LN LEN+20 F100	; Approach P_N; distance P_E to P_N: **LEN+20**

8.6.7 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

> 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 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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**

8.6.8 Departure function DEP LCT

Application

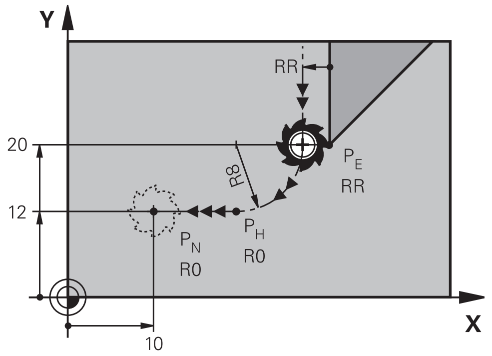
With the NC function **DEP LCT**, the control departs from the contour on a circular path, followed by a tangential straight line to the last contour element.

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 218

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 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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**

8.7 Approach and departure functions with polar coordinates

8.7.1 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.

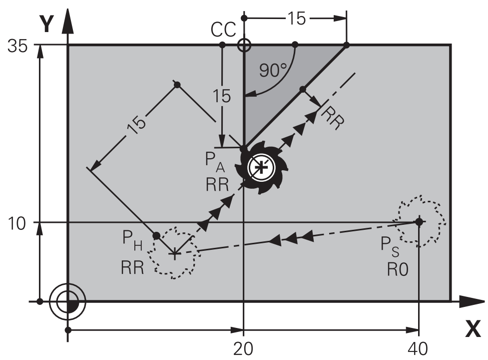
Related topics

- **APPR LT** with Cartesian coordinates
  Further information: "Approach function APPR LT", Page 195

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.7.2 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.

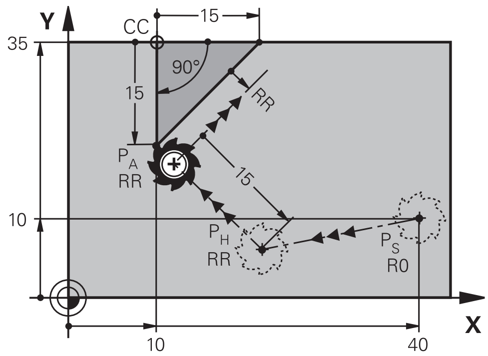
Related topics

- **APPR LN** with Cartesian coordinates
  Further information: "Approach function APPR LN", Page 198

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.7.3 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.

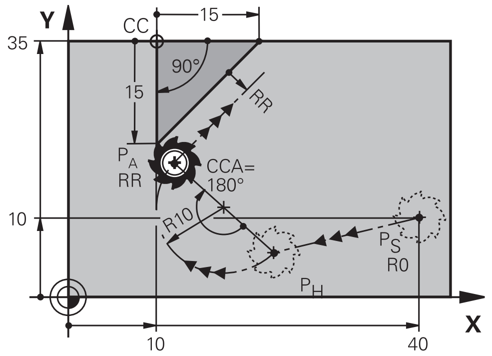
Related topics

- **APPR CT** with Cartesian coordinates
  Further information: "Approach function APPR CT", Page 200

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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

> 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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.7.4 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 202

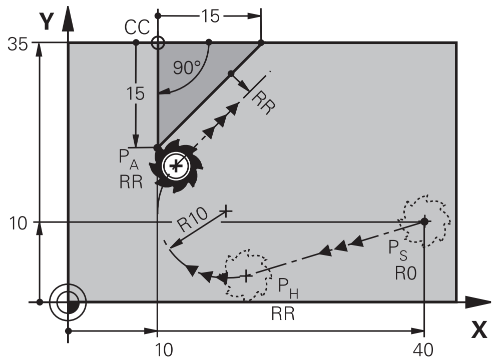
Requirement

- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 181

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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

8.7.5 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.

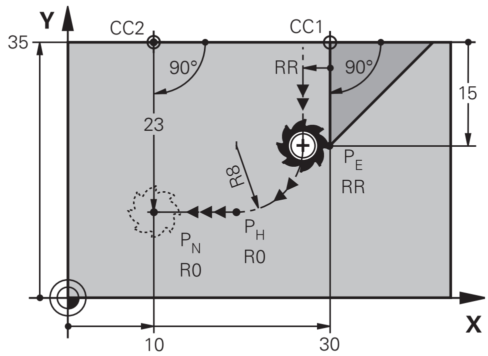
Related topics

- **DEP LCT** with Cartesian coordinates
  Further information: "Departure function DEP LCT", Page 207

Requirement

- Pole **CC**
  You must define a pole **CC** before programming with polar coordinates.
  Further information: "Polar coordinate datum at pole CC", Page 181

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 149 Fixed or variable number Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 437 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 121

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**

9

Programming Techniques

9.1 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 226
- Jumps with conditions as if-then decisions.
  Further information: "The Jump commands folder", Page 500

Description of function

The label **LBL** is used for defining the machining steps for subprograms and program section repeats.

The control offers the following keys and icons in connection with labels:

Key or icon	Function
LBL SET	Create **LBL**
LBL CALL	Call **LBL**: Jump to the label in the NC program
	In case of **LBL** number: Enter the next free number automatically

Defining a label with LBL SET

The **LBL SET** function defines a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by 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 222

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

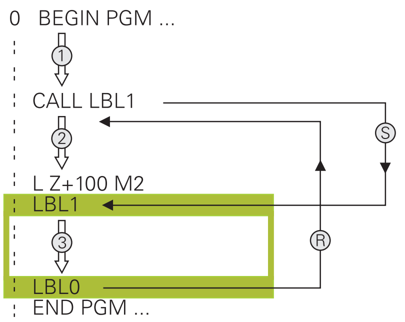
To navigate to this function:

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



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:

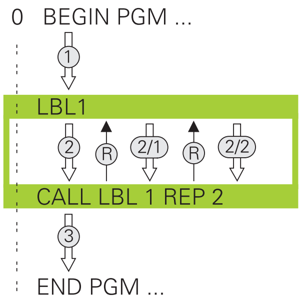
- A subprogram cannot call itself
- **CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).
- Write subprograms after the NC block with M2 or M30

  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: User's Manual for Setup and Program Run

Program-section repeats



A program section repeat allows repeating a part of an NC program any number of times (e.g., contour machining with incremental infeed).

A program section repeat starts with a **LBL** label and ends after the last programmed repetition **REP** of the label call **CALL LBL**.

The control executes the NC program as follows:

1. The control executes the NC program up to the **CALL LBL** function.

   In this process, the control already executes the program section once because the program section to be repeated is positioned ahead of the **CALL LBL** function.
2. The control jumps to the beginning of the program section repeat **LBL**.
3. The control repeats the program section as many times as programmed under **REP**.
4. After that, the control continues executing the NC program.

The following conditions apply to program section repeats:

- Program the program section repeat before the end of the program with **M30** or **M2**.
- No **LBL 0** can be defined with a program section repeat.
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

The control displays information about the active program section repeat on the **LBL** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Notes

- The control displays the NC function **LBL SET** in the structure by default.

  Further information: "The Structure column in the Program workspace", Page 610
- 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> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

9.2 Selection functions

9.2.1 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 226
	Select a datum table with **SEL TABLE**	Page 257
	Select a point table with **SEL PATTERN**	See the User's Manual for Machining Cycles
	Select a contour program with **SEL CONTOUR**	See the User's Manual for Machining Cycles
	Select an NC program with **SEL PGM**	Page 228
	Call the last selected file with **CALL SELECTED PGM**	Page 228
CYC	Select any NC program with **SEL CYCLE** as a machining cycle	See the User's Manual for Machining Cycles
	Select a correction table with **SEL CORR-TABLE**	Page 329
	Open the file with **OPEN FILE**	Page 369
	Link multiple contours with **CONTOUR DEF**	

9.2.2 Call the NC program with CALL PGM

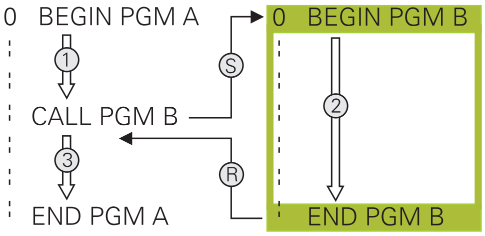
Application

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: User's Manual for Machining Cycles
- Program call following selection
  Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 228
- Executing multiple NC programs as a job list
  Further information: "Pallet Machining and Job Lists", Page 653

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 501

  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 228
- 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.

9.2.3 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 226

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 501

  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 509
- 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.

9.3 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 122
- Mark and copy NC blocks with the context menu

  Further information: "Context menu", Page 618
- Call NC programs unchanged

  Further information: "Call the NC program with CALL PGM", Page 226

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 621
- 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

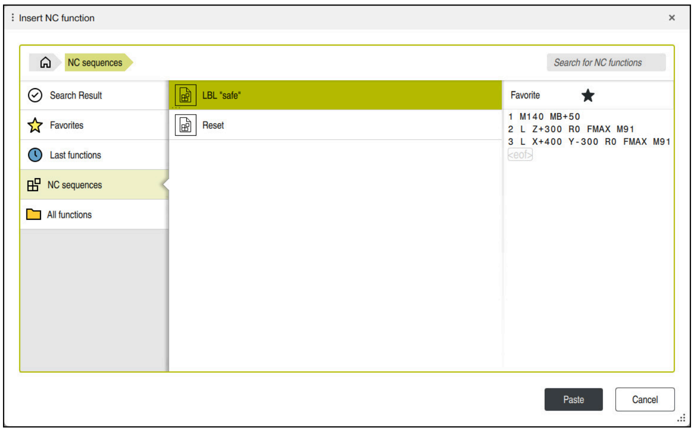
If you create an NC sequence with the **Create NC sequence** button, then the control opens the **Save NC sequence** window.

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 622
- 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: User's Manual for Setup and Program Run
- 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.

9.4 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 224
- Program section repeats

  Further information: "Program-section repeats", Page 225
- Calling a separate NC program

  Further information: "Selection functions", Page 226

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

9.4.1 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.

10

Coordinate Transformation

10.1 Reference systems

10.1.1 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 238
B-CS	Basic coordinate system basic coordinate system	Page 241
W-CS	Workpiece coordinate system workpiece coordinate system	Page 243
WPL-CS	Working plane coordinate system working plane coordinate system	Page 244
I-CS	Input coordinate system input coordinate system	Page 248
T-CS	Tool coordinate system tool coordinate system	Page 249

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.

10.1.2 Basics of coordinate systems

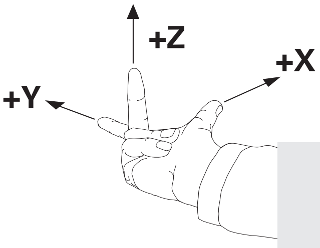
Types of coordinate systems

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.

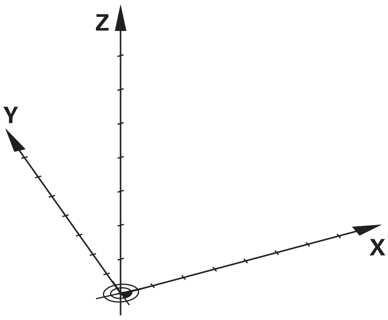
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

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**.



10.1.3 Machine coordinate system M-CS

Application

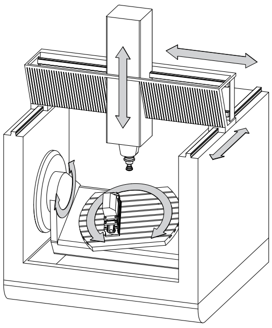
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

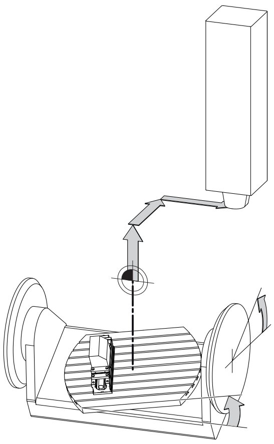
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate 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: User's Manual for Setup and Program Run

  > 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 256
- Axis-specific shifts in the rotary and parallel axes using the **TRANS DATUM** function

  Further information: "Datum shift with TRANS DATUM", Page 259

> The machine manufacturer can also define further transformations.
> **Further information:** "Note", Page 240

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 441

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 669

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**.

10.1.4 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**.

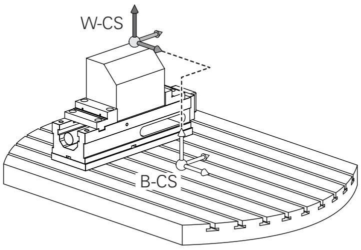
Transformations in the basic coordinate system B-CS

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: User's Manual for Setup and Program Run



The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset table in accordance with the machine.

Further information: "Note", Page 242

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

10.1.5 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.

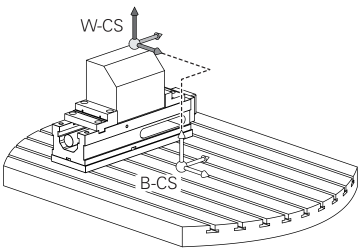
Description of function

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: User's Manual for Setup and Program Run



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 259
- Columns **X**, **Y**, **Z** of the datum table before tilting the working plane

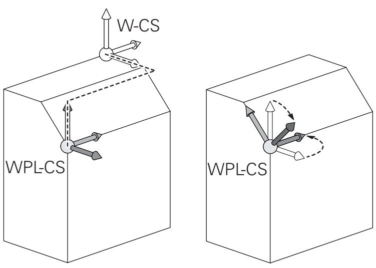
  Further information: "Datum table", Page 256
- The **TRANS MIRROR** function or Cycle **8 MIRRORING** before tilting the working plane with spatial angles

  Further information: "Mirroring with TRANS MIRROR", Page 261
  Further information: User's Manual for Machining Cycles
- **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 269

> 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 248
- 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 244
- The result of transformations built upon each other depends on the programming sequence.

10.1.6 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 248

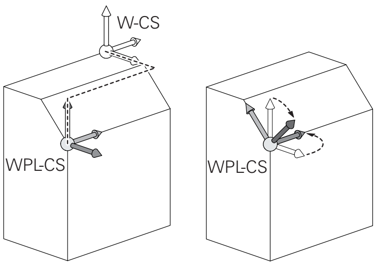
Description of function

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 243

If no transformations are defined in the **W-CS**, then the position and orientation of the **W-CS** and **WPL-CS** are identical.

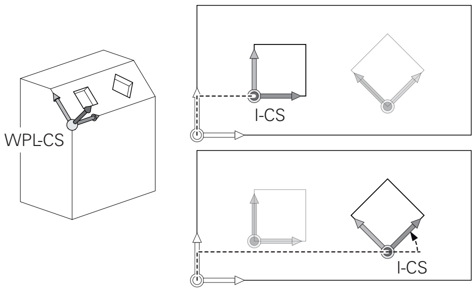


Transformations in the working plane coordinate system WPL-CS

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 259
- The **TRANS MIRROR** function or Cycle **8 MIRRORING**
  Further information: "Mirroring with TRANS MIRROR", Page 261
  Further information: User's Manual for Machining Cycles
- The **TRANS ROTATION** function or Cycle **10 ROTATION**
  Further information: "Rotations with TRANS ROTATION", Page 264
  Further information: User's Manual for Machining Cycles
- The **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
  Further information: "Scaling with TRANS SCALE", Page 265
  Further information: User's Manual for Machining Cycles
- Cycle **26 AXIS-SPECIFIC SCALING**
  Further information: User's Manual for Machining Cycles
- The **PLANE RELATIV** function (#8 / #1-01-1)
  Further information: "PLANE RELATIV", Page 295

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 248
- 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**.

10.1.7 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.

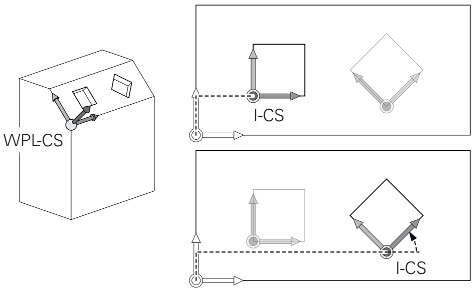
Description of function

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 244

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 249

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 244

10.1.8 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 104

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 141

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 248

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 441

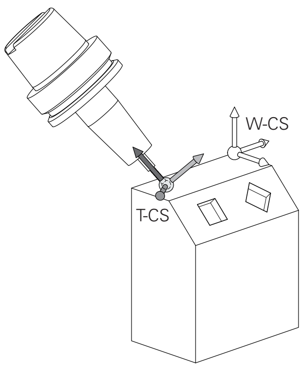
The orientation of the **T-CS** in most cases is identical to that of the **I-CS**.

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 460
- The **FUNCTION TCPM** function (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315



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 463

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 324
- Compensation values from the tool call

  Further information: "Tool compensation for tool length and tool radius", Page 324
- Values of the compensation tables ***.tco**

  Further information: "Tool compensation with compensation tables", Page 329
- 3D tool compensation with surface normal vectors (#9 / #4-01-1)

  Further information: "3D tool compensation (#9 / #4-01-1)", Page 333

10.2 NC functions for preset management

10.2.1 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

10.2.2 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: User's Manual for Setup and Program Run
- Workpiece preset has been defined
  Further information: User's Manual for Setup and Program Run

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: User's Manual for Machining Cycles
- If the pallet preset changes, you need to reset the workpiece preset.

  Further information: "Pallet preset table", Page 669
- 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.

10.2.3 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: User's Manual for Setup and Program Run
- Workpiece preset has been defined

  Further information: User's Manual for Setup and Program Run

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

10.2.4 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: User's Manual for Setup and Program Run
- Workpiece preset has been defined
  Further information: User's Manual for Setup and Program Run

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 236

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_OFFS**, **Z_OFFS**, **A_OFFS**, **B_OFFS**, **C_OFFS**, **U_OFFS**, **V_OFFS**, **W_OFFS**	Compensation value for the offsets, referenced to the machine datum Optional syntax element

10.3 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 694
- Editing a datum table during a program run
  Further information: User's Manual for Setup and Program Run
- Preset table
  Further information: User's Manual for Setup and Program Run

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

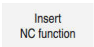
A datum table can be activated manually for the **Program Run** operating mode.

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**.

10.3.1 Activating the datum table in the NC program

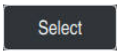
To activate a datum table in the NC program:



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

- Select **SEL TABLE**
- > The control opens the action bar.

- Select **Selection**
- > A file selection window opens.
- Select datum table



- Select **Select**

If the datum table is not stored in the same directory as the NC program, the complete path name must be defined. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 113

> 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

10.4 NC functions for coordinate transformation

10.4.1 Overview

The control provides the following **TRANS** functions:

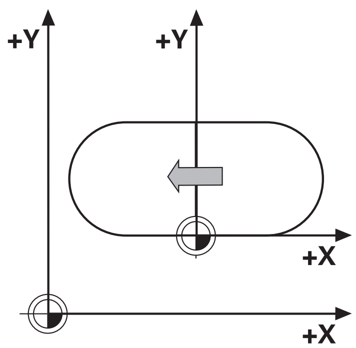
Syntax	Meaning	Further information
TRANS DATUM	Shift the workpiece datum	Page 259
TRANS MIRROR	Mirror an axis	Page 261
TRANS ROTATION	Rotation about the tool axis	Page 264
TRANS SCALE	Scale contours and positions	Page 265
TRANS RESET	Reset the coordinate transformation	Page 266

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 248



Related topics

- Coordinate transformation cycles

  Further information: User's Manual for Machining Cycles
- **PLANE** functions (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 269
- Reference systems

  Further information: "Reference systems", Page 236

10.4.2 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 694
- Activating the datum table

  Further information: "Activating the datum table in the NC program", Page 257
- Machine presets

  Further information: "Presets in the machine", Page 104

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: User's Manual for Setup and Program Run

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 257

The control displays the datum shift and the path to the datum table on the **TRANS** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

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 104
- 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: User's Manual for Setup and Program Run
- 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 236

10.4.3 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

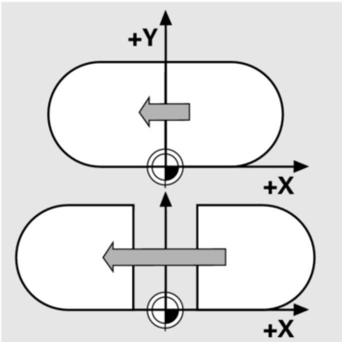
- Cycle **8 MIRRORING**
  Further information: User's Manual for Machining Cycles

Description of function

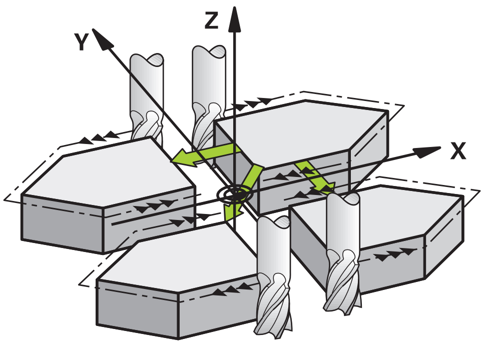
Mirroring is a modal function that in effect as soon as it has been defined in the NC program.

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 104



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)).

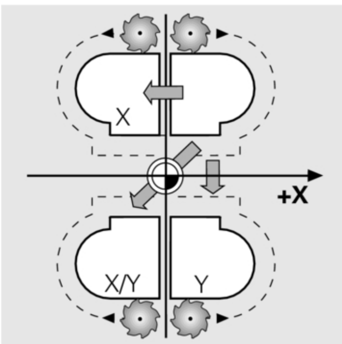


Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **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 102

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: User's Manual for Setup and Program Run

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 130
- If you execute mirroring with **TRANS MIRROR** or Cycle **8 MIRRORING**, then the control overwrites the current mirroring.

  Further information: User's Manual for Machining Cycles

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 243

10.4.4 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: User's Manual for Machining Cycles

Description of function

Rotation is a modal function that is in effect as soon as it has been defined in the NC program.

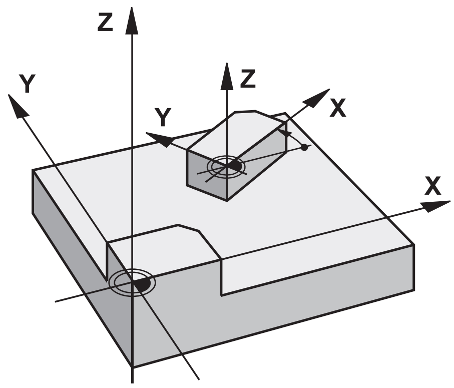
The control rotates machining in the working plane about the active workpiece datum.

Further information: "Presets in the machine", Page 104

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 102



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: User's Manual for Setup and Program Run

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 130
- 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: User's Manual for Machining Cycles

10.4.5 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: User's Manual for Machining Cycles

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:

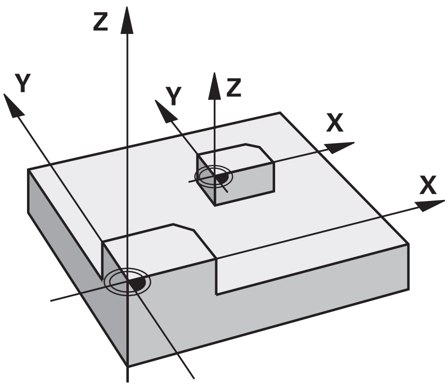
- Workpiece datum at the center of the contour:

  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 104



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: User's Manual for Setup and Program Run

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 130
- 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: User's Manual for Machining Cycles
- 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.

10.4.6 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: User's Manual for Machining Cycles

Description of function

The control resets the following simple coordinate transformations:

Coordinate transformation	Syntax	Further information
Datum shift	**TRANS DATUM**	Page 259
Mirroring	**TRANS MIRROR**	Page 261
	Cycle **8 MIRRORING**	See the User's Manual for Machining Cycles
Rotation	**TRANS ROTATION**	Page 264
	Cycle **10 ROTATION**	See the User's Manual for Machining Cycles
Scaling	**TRANS SCALE**	Page 265
	Cycle **11 SCALING FACTOR**	See the User's Manual for Machining Cycles
	Cycle **26 AXIS-SPECIFIC SCALING**	See the User's Manual for Machining Cycles

> 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

10.5 Tilting the working plane (#8 / #1-01-1)

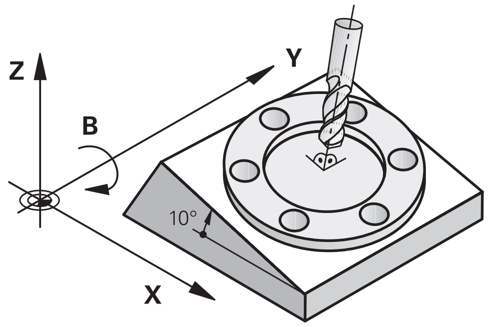
10.5.1 Fundamentals

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 244



Two functions are available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application

  Further information: User's Manual for Setup and Program Run
- 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 269

> 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 241

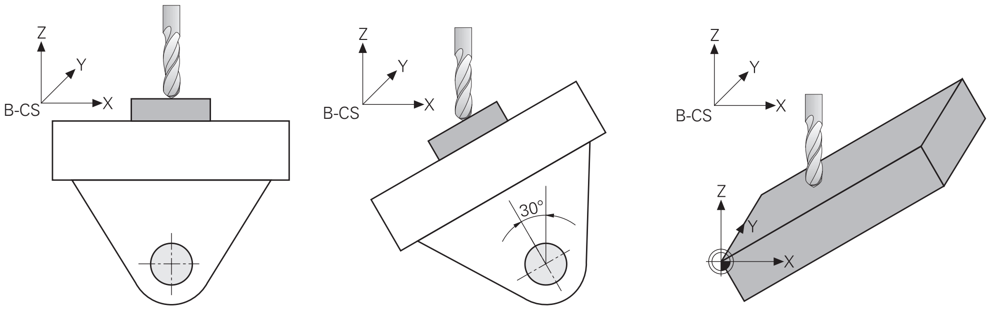
When tilting the working plane, the control moves the machine axes according to the kinematics.

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**.

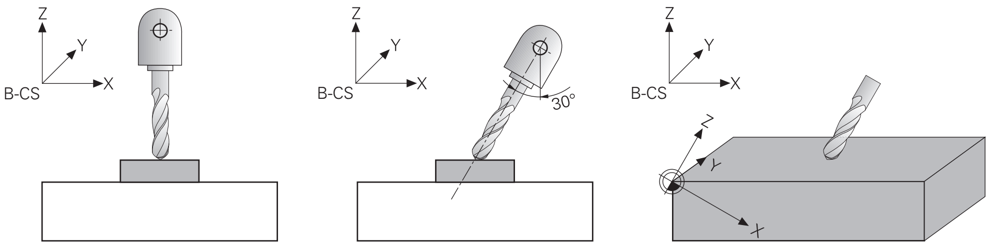
  Further information: "Working plane coordinate system WPL-CS", Page 244



- 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 244



10.5.2 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 425
- Adopting a tilted working plane in the **Manual** operating mode with the **3-D rotation** window

  Further information: User's Manual for Setup and Program Run

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 236

> 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 259

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 271

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 303

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 306

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 310

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: User's Manual for Setup and Program Run

> When deactivating or resetting the tilting function correctly, the icon indicating the tilted working plane must disappear.
>
> **Further information:** "PLANE RESET", Page 299

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: User's Manual for Setup and Program Run

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 274
PROJECTED	Defines the working plane by means of two projection angles and one rotation angle	Page 280
EULER	Defines the working plane by means of three Euler angles	Page 284
VECTOR	Defines the working plane by means of two vectors	Page 287
POINTS	Defines the working plane by means of the coordinates of three points	Page 290
RELATIV	Defines the working plane by means of a single spatial angle with incremental effect	Page 295
AXIAL	Defines the working plane by means of a maximum of three absolute or incremental axis angles	Page 300
RESET	Resets tilting of the working plane	Page 299

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
```

> 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.

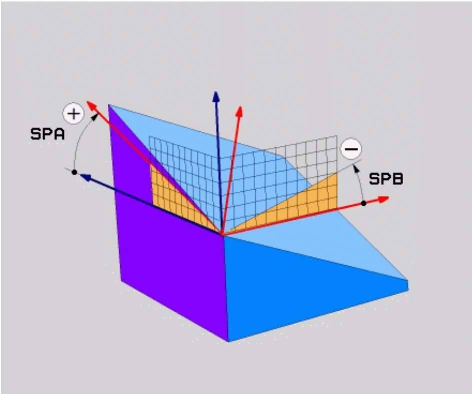
> Spatial angles are the most frequently used definition option for a working plane. The definition is not machine-specific, meaning that it is independent of the rotary axes actually present.

Related topics

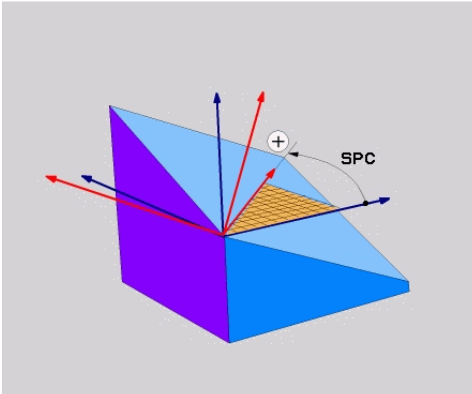
- Defining a single spatial angle with incremental effect
  Further information: "PLANE RELATIV", Page 295
- Entering the axis angle
  Further information: "PLANE AXIAL", Page 300

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.



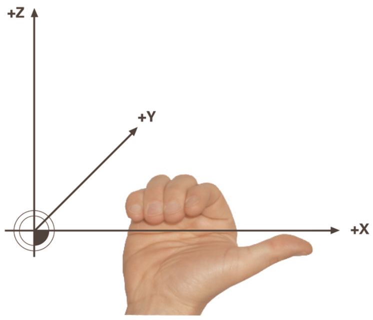
Spatial angles **SPA** and **SPB**



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**.

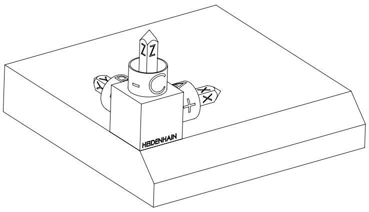
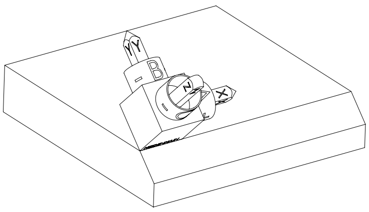
Further information: "Notes", Page 278

> 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 295

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 278
- **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 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 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 Optional syntax element

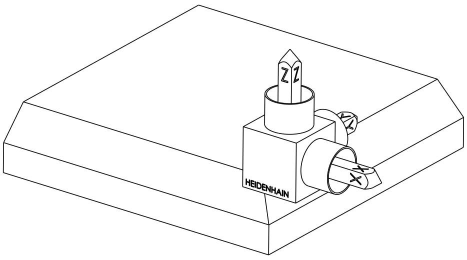
Notes

Comparison of views - Example: chamfer

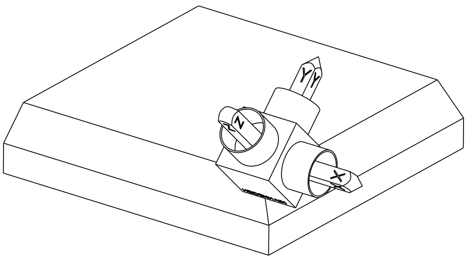
Example

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+90 TURN MB MAX FMAX SYM- TABLE ROT
```

View A-B-C



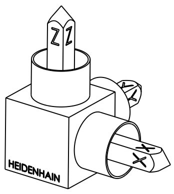
Initial state



SPA+45

Orientation of tool axis **Z**

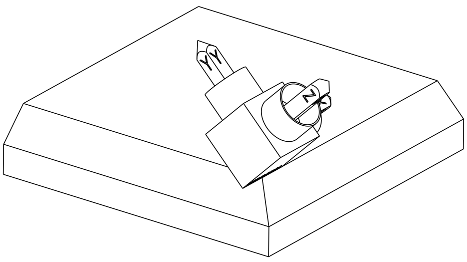
Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**



SPB+0

Rotation around the Y axis of the non-tilted **W-CS**

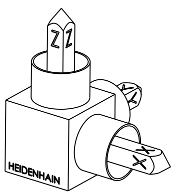
No rotation with value 0



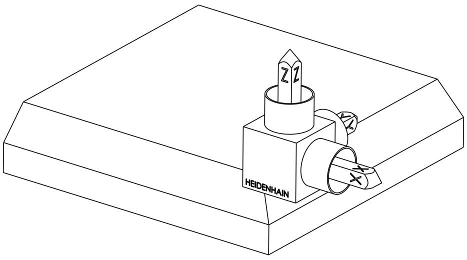
SPC+90

Orientation of main axis **X**

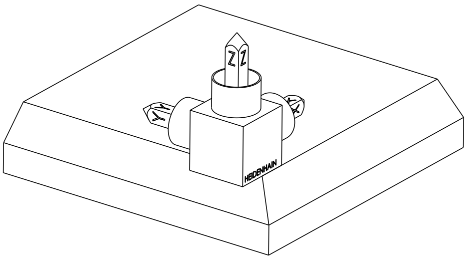
Rotation around the Z axis of the non-tilted **W-CS**



View C-B-A



Initial state



SPC+90

Orientation of main axis **X**

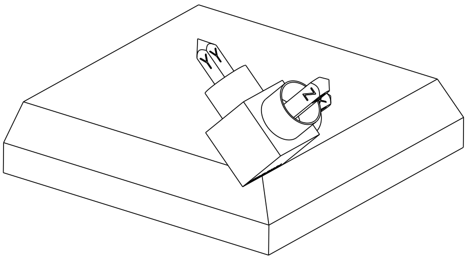
Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane

---

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

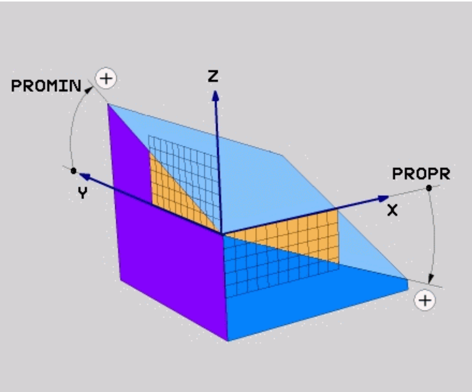
Use the **PLANE PROJECTED** function to define the working plane by two projection angles. Use an additional rotation angle to optionally align the X axis in the tilted working plane.

Description of function

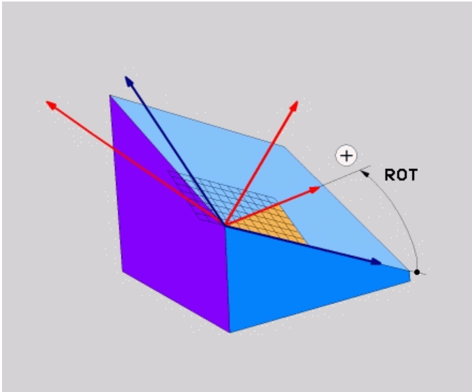
Projection angles define a working plane through two independent angles in the working planes **ZX** and **YZ** of the non-tilted working plane coordinate system **W-CS**.

Further information: "Designation of the axes of milling machines", Page 102

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.

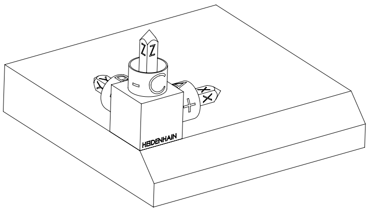
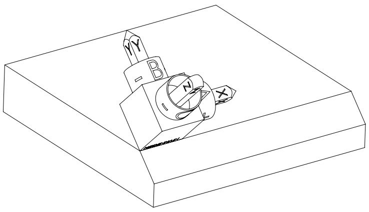
The projection angles of non-rectangular workpieces can be obtained by imagining the working planes **ZX** and **YZ** as transparent panels with angle scales. When viewing the workpiece from the front through the **ZX** plane, the difference between the X axis and the workpiece edge equals the projection 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 283

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

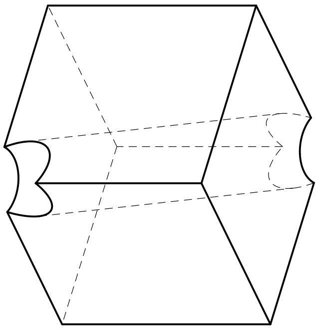
```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE
ROT
```

The NC function includes the following syntax elements:

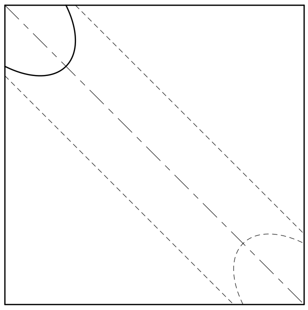
Syntax element	Meaning
PLANE PROJECTED	Syntax initiator for the working plane definition by means of two projection angles and one rotation angle
PROPR	Angle in working plane **ZX**, i. e. around the Y axis of the workpiece coordinate system **W-CS** Input: **-89.999999...+89.9999**
PROMIN	Angle in the 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 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 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 Optional syntax element

Notes

Procedure in case of hidden workpiece edges, using the example of a diagonal hole



Cube with a diagonal hole

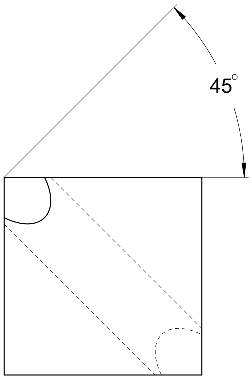


Front view, meaning projection on the **ZX** working plane

Example

```
11 PLANE PROJECTED PROPR-45  PROMIN+45 ROT+0 TURN MB MAX FMAX SYM-
   TABLE ROT
```

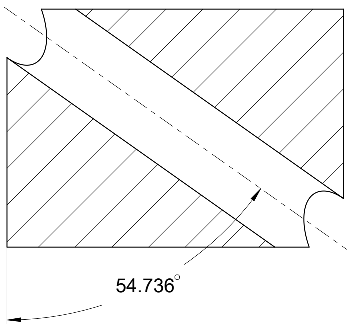
Comparison of projection and spatial angles



When imagining the workpiece to be transparent, the projection angles are easy to find.

Both projection angles are 45°.

> **i** When defining the algebraic sign, ensure that the working plane is perpendicular to the center axis of the hole.



When defining the working plane by using spatial angles, the spatial diagonal must be considered.

The full section along the hole axis shows that the axis does not form an isosceles triangle with the lower and the left workpiece edge. This is why e. g. a spatial angle **SPA+45** produces an incorrect result.

Definition

Abbreviation	Definition
PROPR	Main plane
PROMIN	Minor plane
ROT	Angle of rotation

PLANE EULER

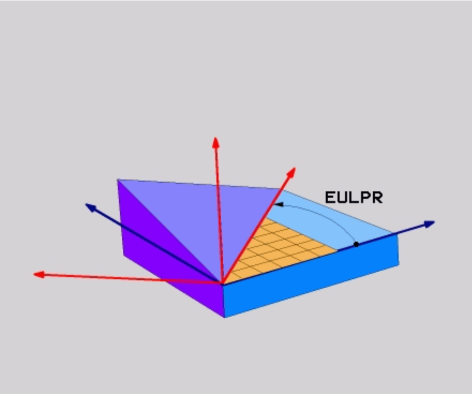
Application

Use the **PLANE EULER** function to define the working plane by three Euler angles.

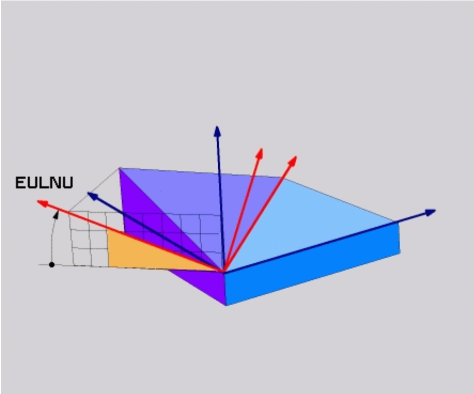
Description of function

Euler angles define a working plane as three rotations layered on top of one another, starting from the non-tilted workpiece coordinate system **W-CS**.

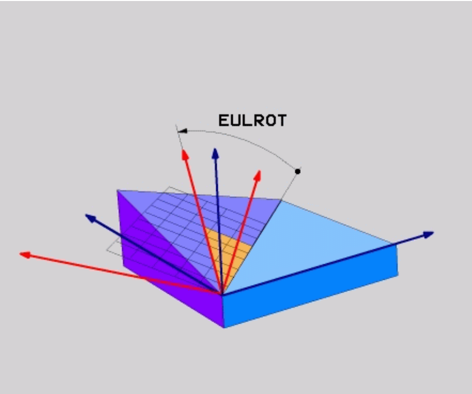
Use the third Euler angle to optionally align the tilted X axis.



Euler angle **EULPR**



Euler angle **EULNU**



Euler angle **EULROT**

All three angles must be defined even if one or several angles equals 0.

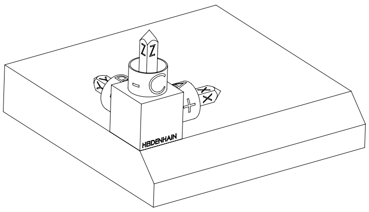
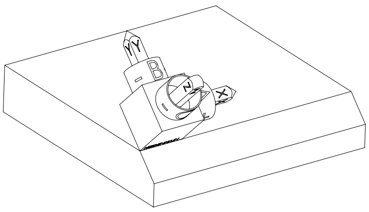
At first, the rotations layered on top of one another happen around the non-tilted Z axis, then around the tilted X axis and finally around the tilted Z axis.

> This view equals three **PLANE RELATIV** functions programmed one-by-one, first with **SPC**, then with **SPA** and finally with **SPC** again.
>
> **Further information:** "PLANE RELATIV", Page 295
>
> 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 274
>
> **Further information:** "Rotations with TRANS ROTATION", Page 264

Application example

Example

```
11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined Euler angle **EULNU**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **EULNU** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following Euler angles:
>
> - **EULPR+90**, **EULNU45** and **EULROT0** for the second chamfer
> - **EULPR+180**, **EULNU45** and **EULROT0** for the third chamfer
> - **EULPR+270**, **EULNU45** and **EULROT0** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

Example

```
11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE EULER	Syntax initiator for the working plane definition by means of three Euler angles
EULPR	Rotation around the Z axis of the workpiece coordinate system **W-CS** Input: **-180.000000...+180.000000**
EULNU	Rotation around the X axis of the tilted working plane coordinate system **WPL-CS** Input: **0...180.000000**
EULROT	Rotation around the Z axis of the tilted **WPL-CS** Input: **0...360.000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 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.

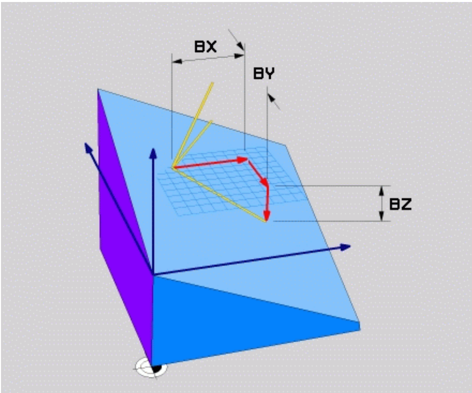
Related topics

- Output formats of NC programs

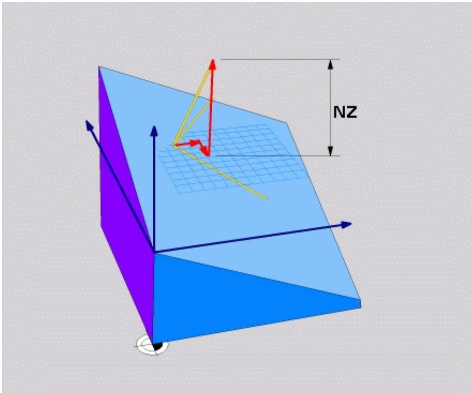
  Further information: "Output formats of NC programs", Page 423

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.

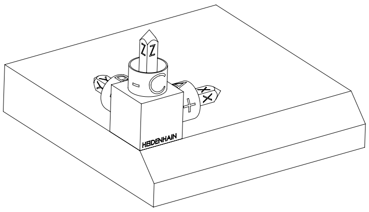
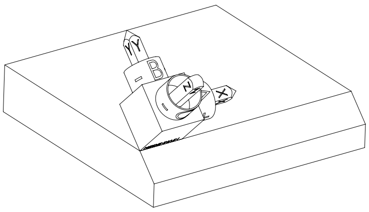
> There is no need to enter a normalized vector. The drawing dimensions or any values which will not alter the ratio between the components can be used.
>
> **Further information:** "Application example", Page 288

The base vector with components **BX**, **BY** and **BZ** defines the direction of the tilted X axis. The normal vector with components **NX**, **NY** and **NZ** defines the direction of the tilted Z axis and therefore indirectly the working plane. The normal vector is perpendicular to the tilted working plane.

Application example

Example

```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined normal vector with the components **NX+0**, **NY-1** and **NZ+1**, the control orients the Z axis of the working plane coordinate system **WPL-CS** to be perpendicular with the chamfer surface. The alignment of the tilted X axis equals the orientation of the non-tilted X axis due to component **BX+1**. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following vector components:

- **BX+0**, **BY+1** and **BZ+0** as well as **NX+1**, **NY+0** and **NZ+1** for the second chamfer
- **BX-1**, **BY+0** and **BZ+0** as well as **NX+0**, **NY+1** and **NZ+1** for the third chamfer
- **BX+0**, **BY-1** and **BZ+0** as well as **NX-1**, **NY+0** and **NZ+1** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE VECTOR	Syntax initiator for the working plane definition by means of two vectors
BX, **BY** and **BZ**	Components of base vector, referenced to the workpiece coordinate system **W-CS**, for orienting the tilted X axis Input: **-99.9999999...+99.9999999**
NX, **NY** and **NZ**	Components of the normal vector, referenced to the **W-CS**, for orienting the tilted Z axis Input: **-99.9999999...+99.9999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 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.

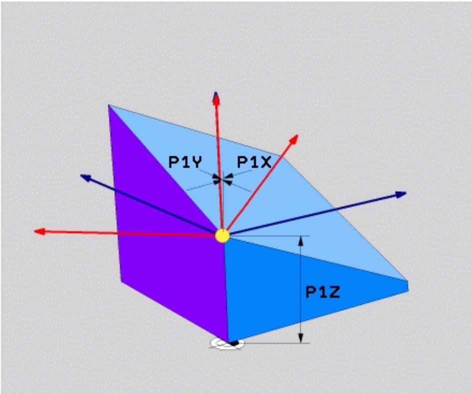
Related topics

- Aligning the plane with touch probe cycle **431 MEASURE PLANE**

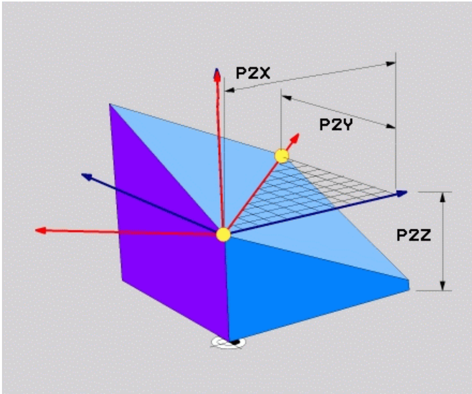
  Further information: User's Manual for Measuring Cycles for Workpieces and Tools

Description of function

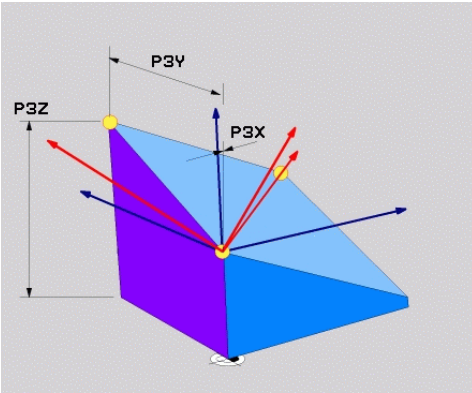
Points define a working plane by using their coordinates in the non-tilted workpiece coordinate system **W-CS**.



First point with coordinates **P1X**, **P1Y** and **P1Z**



Second point with coordinates **P2X**, **P2Y** and **P2Z**



Third point with coordinates **P3X**, **P3Y** and **P3Z**

All nine coordinates must be defined even if one or several coordinates equals 0.

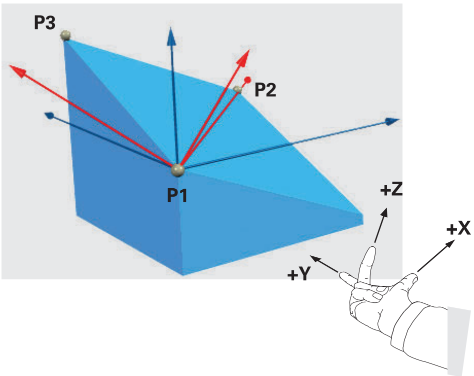
The first point with coordinates **P1X**, **P1Y** and **P1Z** defines the first point of the tilted X axis.

> You can imagine that the first point defines the origin of the tilted X axis and therefore the point serving for orientation of the working plane coordinate system **WPL-CS**.
>
> Ensure that the definition of the first point will not shift the workpiece datum. If the coordinates of the first point are to be programmed with the value 0, the workpiece datum may have to be shifted to that position before.

The second point with coordinates **P2X**, **P2Y** and **P2Z** defines the second point of the tilted X axis and consequently its orientation.

> The orientation of the tilted Y axis in the defined working plane results automatically because both axes are perpendicular to one another.

The third point with coordinates **P3X**, **P3Y** and **P3Z** defines the slope of the tilted working plane.



To direct the positive tool axis direction away from the workpiece, the following conditions apply to the position of the three points:

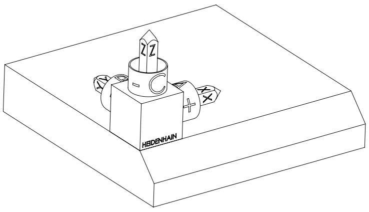
- Point 2 is to the right of point 1
- Point 3 is above the connecting lines between points 1 and 2

Application example

Example

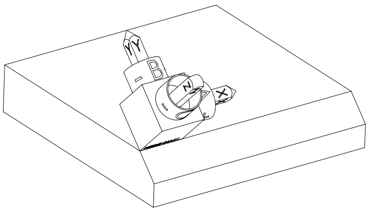
```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the first two points **P1** and **P2**, the control orients the X axis of the **WPL-CS**.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

P3 defines the slope of the tilted working plane.

The orientations of the tilted Y and Z axes result automatically because all axes are perpendicular to one another.

> The drawing dimensions or any values which will not alter the ratio between the entered values can be used.
>
> In the example, **P2X** may also be defined by the workpiece width **+100**. **P3Y** and **P3Z** can also be programmed by using the chamfer width **+10**.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following points:
>
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y+1**, **P2Z+0** and **P3X-1**, **P3Y+0**, **P3Z+1** for the second chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X-1**, **P2Y+0**, **P2Z+0** and **P3X+0**, **P3Y-1**, **P3Z+1** for the third chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y-1**, **P2Z+0** and **P3X+1**, **P3Y+0**, **P3Z+1** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
    TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE POINTS	Syntax initiator for the working plane definition by means of three points
P1X, **P1Y** and **P1Z**	Coordinates of the first point of the tilted X axis, referenced to the workpiece coordinate system **W-CS** Input: **-999999999.999999...+999999999.999999**
P2X, **P2Y** and **P2Z**	Coordinates of the second point, referenced to the **W-CS** for orienting the tilted X axis Input: **-999999999.999999...+999999999.999999**
P3X, **P3Y** and **P3Z**	Coordinates of the third point, referenced to the **W-CS** for inclining the tilted working plane Input: **-999999999.999999...+999999999.999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 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 236

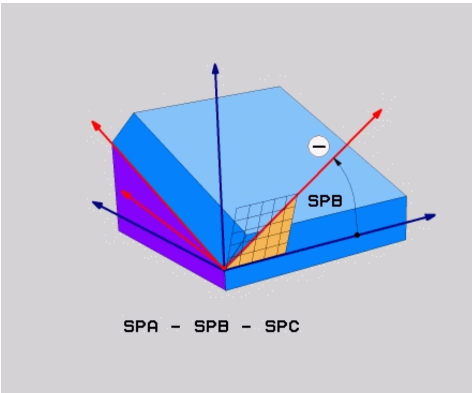
Description of function

A relative spatial angle defines a working plane as a rotation in the active reference system.

When the working plane is not tilted, the defined spatial angle is referenced to the non-tilted workpiece coordinate system **W-CS**.

When the working plane is tilted, the defined spatial angle is referenced to the working plane coordinate system **WPL-CS**.

PLANE RELATIV allows, 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.

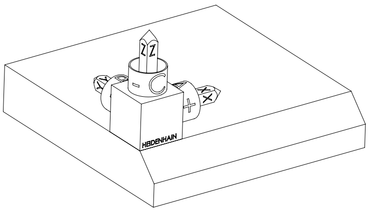
If you want to return the working plane that was active before the **PLANE RELATIV** function, define another **PLANE RELATIV** function with the same angle, but with the opposite algebraic sign.

Application example

Example

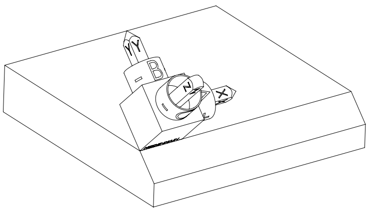
```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

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 298

Input

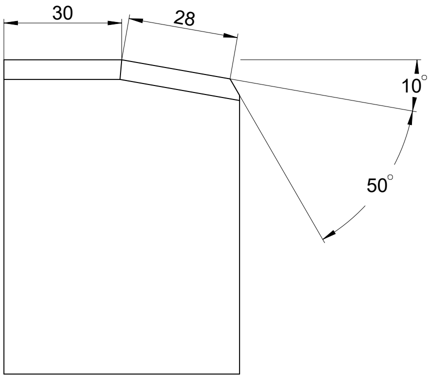
```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

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 303
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 306 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 310 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: User's Manual for Setup and Program Run
- 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 303

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 303

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: User's Manual for Setup and Program Run

  > The status display allows checking the desired status of the tilting situation.
  >
  > **Further information:** "Status display", Page 271

- 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 274

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.

> 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

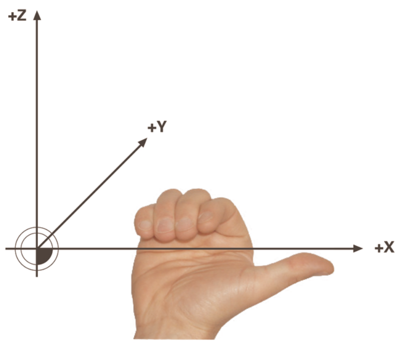
  Further information: "PLANE SPATIAL", Page 274

Description of function

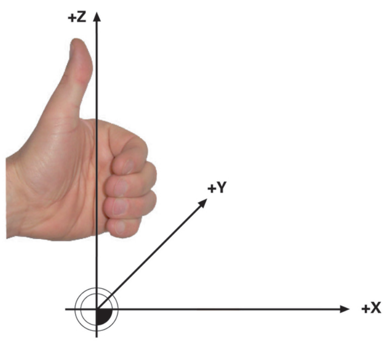
Axis angles define both the orientation of the working plane as well as the nominal coordinates of the rotary axes.

The axis angles must correspond to the axes present on the machine. If you try to program axis angles for rotary axes that do not exist on the machine, the control will generate an error message.

As the axis angles depend on the kinematics, a distinction must be made between the head and the table axes as far as the algebraic signs are concerned.



Extended right-hand rule for head rotary axes



Extended left-hand rule for table rotary axes

The thumb of the hand in question points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

Bear in mind that when working with rotary axes layered on top of one another, the positioning of the first rotary axis will also modify the position of the second rotary axis.

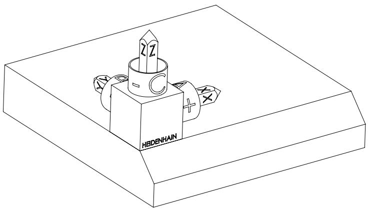
Application example

The example below applies to a machine with AC table kinematics whose two rotary axes are perpendicular and layered on top of one another.

Example

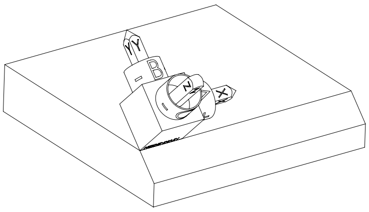
```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

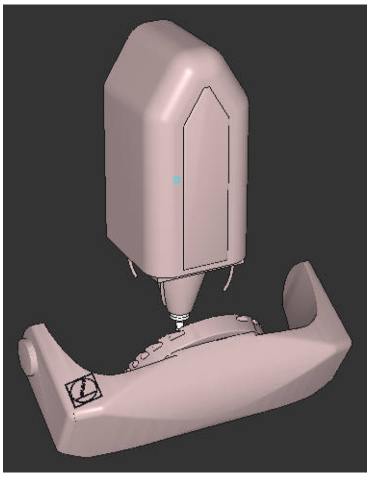
Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis





Using the defined axis angle **A**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by angle **A** is around the non-tilted X axis.

> To position the tool perpendicular to the chamfer surface, table rotary axis A must tilt to the rear.
>
> In accordance with the extended left-hand rule for table axes, the algebraic sign of the A axis value must be positive.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following axis angles:
>
> - **A+45** and **C+90** for the second chamfer
> - **A+45** and **C+180** for the third chamfer
> - **A+45** and **C+270** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE AXIAL	Syntax initiator for the working plane definition using one to three axis angle
A	When an A axis is available, nominal position of the A rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
B	When a B axis is available, nominal position of the B rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
C	When a C axis is available, nominal position of the C rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 303

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 243
- Without the use of a CAM system, **PLANE AXIAL** is convenient only with rotary axes positioned at right angles.

Rotary axis positioning

Application

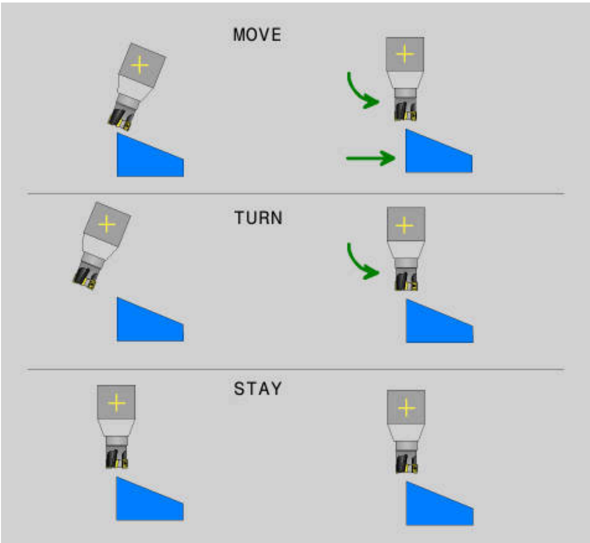
The type of rotary axis positioning defines how the control tilts the rotary axes to the calculated axis values.

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 304
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 304
STAY	The control does not position any axes. **Further information:** "Rotary axis positioning with STAY", Page 305

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!

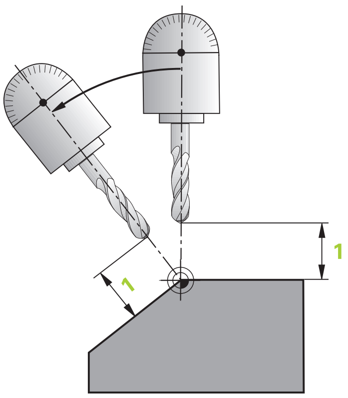
The center of rotation is in the tool axis. In the case of large tool diameters, the tool may plunge into the material during tilting. During the tilting movement, there is a risk of collision!

- ▶ 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.

> 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

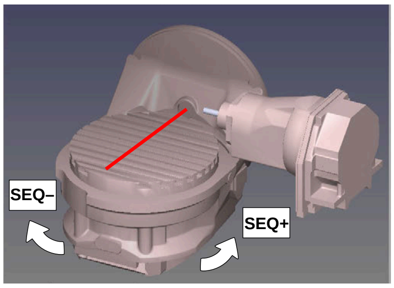
SYM (SEQ) allows selecting the desired option from several tilting solutions.

> 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.

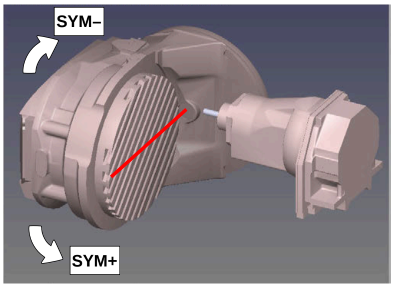
Description of function

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 307
SEQ	With **SEQ** you select a tilting solution relative to the basic position of the master axis. **Further information:** "Tilting solution SEQ", Page 308



Reference for **SEQ**



Reference for **SYM**

If the solution you have selected with **SYM** (**SEQ**) is not within the machine's range of traverse, then the control displays the **Entered angle not permitted** error message.

The entry of **SYM** or **SEQ** is optional.

If you do not define **SYM** (**SEQ**), then the control determines the solution as follows:

1. Check whether both possible solutions are within the traverse range of the rotary axes
2. Two possible solutions: Based on the current position of the rotary axes, choose the possible solution with the shortest path
3. One possible solution: Choose the only solution
4. No possible solution: Issue the error message **Entered angle not permitted**

Tilting solution SYM

With the **SYM** function, you select one of the possible solutions relative to the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space relative to the symmetry point
- **SYM-** positions the master axis in the negative half-space relative to the symmetry point

As opposed to **SEQ**, **SYM** uses the symmetry point of the master axis as its reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is in the traverse range).

> To determine the symmetry point:
> - Perform **PLANE SPATIAL** with any spatial angle and **SYM+**
> - Save the axis angle of the master axis in a Q parameter (e.g., –80)
> - Repeat the **PLANE SPATIAL** function with **SYM-**
> - Save the axis angle of the master axis in a Q parameter (e.g., –100)
> - Calculate the average value (e.g., –90)
>
>   The average value corresponds to the symmetry point.

Tilting solution SEQ

With the **SEQ** function, you select one of the possible solutions relative to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range relative to the home position
- **SEQ-** positions the master axis in the negative tilting range relative to the home position

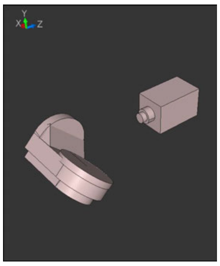
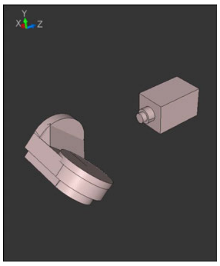
SEQ assumes that the master axis is in its home position (0°). Relative to the tool, the master axis is the first rotary axis, or the last rotary axis relative to the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, then the control automatically uses the closer solution (shorter path). If you need the second possible solution, then you must either pre-position the master axis (in the area of the second possible solution) before tilting the working plane, or work with **SYM**.

Examples

Machine with C rotary axis and A tilting table.
Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Start position	SYM = SEQ	Resulting axis position
None	A+0, C+0	Not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A−45, C−90
None	A+0, C−105	Not prog.	A−45, C−90
None	A+0, C−105	+	A+45, C+90
None	A+0, C−105	–	A−45, C−90
−90 < A < +10	A+0, C+0	Not prog.	A−45, C−90
−90 < A < +10	A+0, C+0	+	Error message
−90 < A < +10	A+0, C+0	–	A−45, C−90

Machine with B rotary axis and A tilting table (limit switches: A +180 and −100).
Programmed function: PLANE SPATIAL SPA-45 SPB+0 SPC+0

SYM	SEQ	Resulting axis position	Kinematics view
+		A−45, B+0	
-		Error message	**No solution in limited range**
	+	Error message	**No solution in limited range**
	-	A−45, B+0	

The position of the symmetry point is contingent on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point changes as well.

Depending on the kinematics, the positive direction of rotation of **SYM** may not correspond to the positive direction of rotation of **SEQ**. Therefore, ascertain the position of the symmetry point and the direction of rotation of **SYM** on each machine before programming.

Transformation types

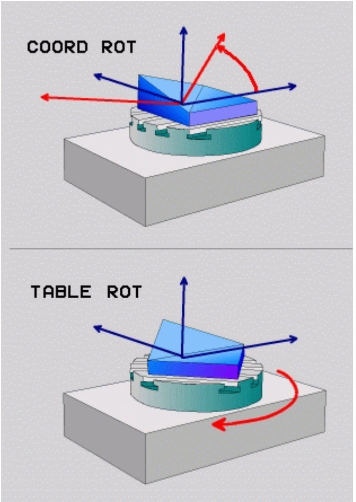
Application

COORD ROT and **TABLE ROT** influence the orientation of the working plane coordinate system **WPL-CS** through the axis position of a free rotary axis.

> Any rotary axis becomes a free rotary axis with the following configuration:
> - The rotary axis has no effect on the tool angle of inclination because the rotary axis and the tool axis are parallel in the tilting situation
> - The rotary axis is the first rotary axis in the kinematic chain starting from the workpiece
>
> The effect of the **COORD ROT** and **TABLE ROT** transformation types therefore depends on the programmed spatial angles and the machine kinematics.

Description of function

The control offers two options.



Option	Meaning
COORD ROT	> The control positions the free rotary axis to 0 > The control orients the working plane coordinate system in accordance with the programmed spatial angle
TABLE ROT	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control orients the free rotary axis in accordance with the programmed spatial angle > The control orients the working plane coordinate system in accordance with the basic coordinate system **TABLE ROT** with: ■ **At least** SPA **or** SPB **unequal to** 0 ■ SPC **equal or unequal to** 0 > The control does not position the free rotary axis. The position prior to tilting the working plane is maintained > Since the workpiece was not positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle

If no free rotary axis arises in a tilting situation, then the **COORD ROT** and **TABLE ROT** transformation types have no effect.

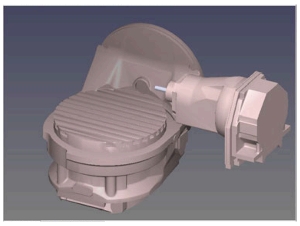
The entry of **COORD ROT** or **TABLE ROT** is optional.

If no transformation type was selected, then the control uses the **COORD ROT** transformation type for the **PLANE** functions

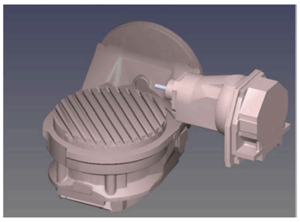
Example

The following example shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

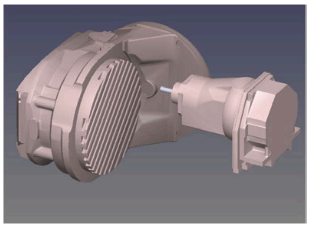
11 L B+45 R0 FMAX	; Pre-position the rotary axis
12 PLANE SPATIAL SPA-90 SPB+20 SPC +0 TURN F5000 TABLE ROT	; Tilt the working plane



Origin



A = 0, B = 45



A = −90, B = 45

> The control positions the B axis to the axis angle B+45
> With the programmed tilting situation with SPA−90, the B axis becomes the free rotary axis
> The control does not position the free rotary axis. The position of the B axis prior to the tilting of the working plane is maintained
> Since the workpiece was not also positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle SPB +20

Notes

- For the positioning behavior with the **COORD ROT** and **TABLE ROT** transformation types, it makes no difference whether the free rotary axis is a table axis or a head axis.
- The resulting axis position of the free rotary axis depends on an active basic rotation, among other factors.
- The orientation of the working plane coordinate system is also dependent on a programmed rotation (e.g., with Cycle **10 ROTATION**).

10.6 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 315
- 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 460
- Tilting the working plane (#8 / #1-01-1)

  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 268
- Presets on the tool

  Further information: "Presets on the tool", Page 141
- Reference systems

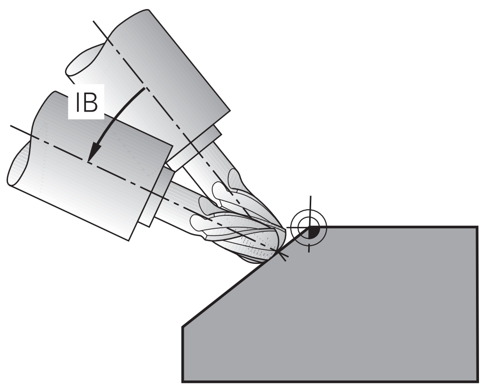
  Further information: "Reference systems", Page 236

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 268

Inclined machining can be implemented using the following functions:

- Incremental traverse of rotary axis

  Further information: "Inclined machining with incremental process", Page 314
- Normal vectors

  Further information: "Inclined machining using normal vectors", Page 314

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
*** - ...**	

10.7 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.

> 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 460
- Tilting the working plane

  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 268
- Presets on the tool

  Further information: "Presets on the tool", Page 141
- Reference systems

  Further information: "Reference systems", Page 236

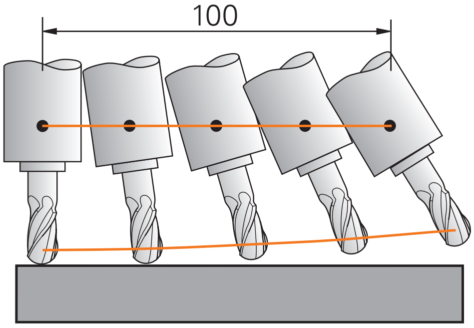
Requirements

- Machine with rotary axes
- Kinematics description

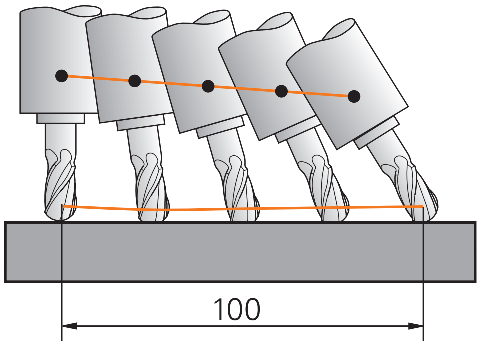
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Software option Advanced Functions Set 2 (#9 / #4-01-1)

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: User's Manual for Setup and Program Run

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 317
AXIS POS or **AXIS SPAT**	Interpretation of programmed rotary axis coordinates **Further information:** "Interpretation of the programmed rotary axis coordinates", Page 318
PATHCTRL AXIS or **PATHCTRL VECTOR**	Interpolation of tool angle of inclination **Further information:** "Interpolation of tool angle of inclination between start and end positions", Page 319
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 320 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 321 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**

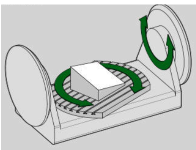
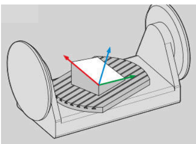
Interpretation of the programmed feed rate

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.

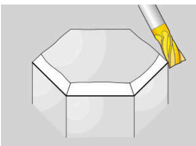
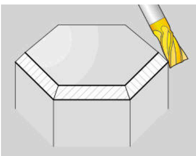
Interpretation of the programmed rotary axis coordinates

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 248

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:** User's Manual for Machining Cycles **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 337 **Further information:** "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 344
 **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.

> To obtain continuous movement, define Cycle **32** with a **tolerance for rotary axes**.
> **Further information:** User's Manual for Machining Cycles

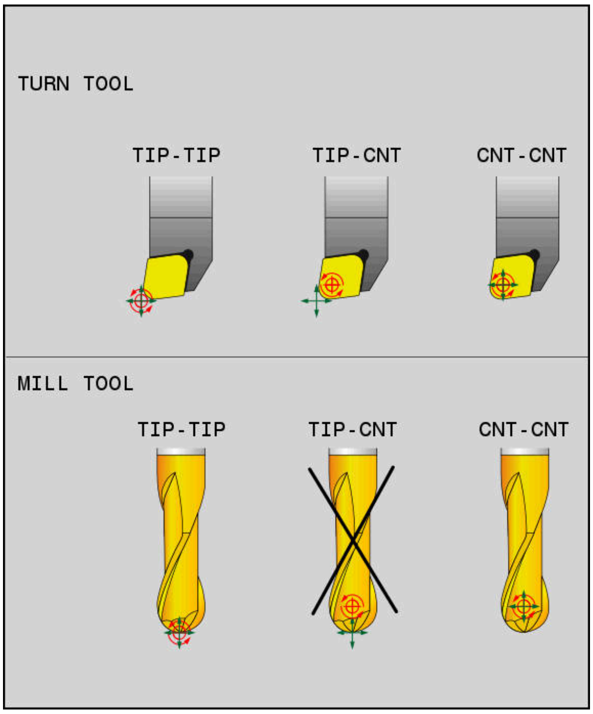
Selection of tool location point and tool rotation point

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 326 If you use **REFPNT CENTER-CENTER** to program pocket milling cycles, the control generates an error message. **Further information:** User's Manual for Machining Cycles

Further information: "Presets on the tool", Page 141

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).

> 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: "Straight line LN", Page 334
- 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 463

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: User's Manual for Setup and Program Run

- 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 243
- 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.

11

Compensations

11.1 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 153

Related topics

- Tool radius compensation

  Further information: "Tool radius compensation", Page 326
- Tool compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 329

Description of function

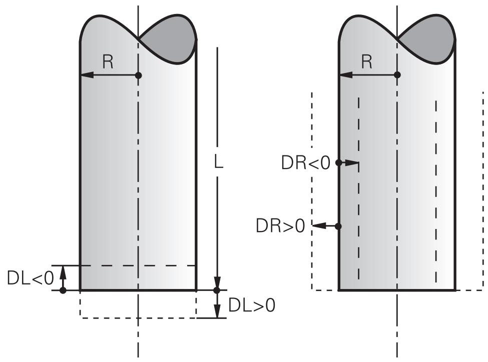
The control distinguishes between two types of delta values:

- 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: User's Manual for Setup and Program Run
- 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 144



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 144

Further information: "Tool compensation with compensation tables", Page 329

Delta values within a tool call can also be defined by using variables.

Further information: "Tool data within variables", Page 326

Tool length compensation

The control takes the tool length compensation into account as soon as a tool is called. The control performs tool length compensation only on tools of length L>0.

In tool length compensation, the control takes delta values from the tool table and the NC program into account.

Active tool length = **L** + $\mathbf{DL}_{TAB}$ + $\mathbf{DL}_{Prog}$

L:	Tool length **L** from the tool table
$\mathbf{DL}_{TAB}$:	Tool length delta value **DL** from the tool table
$\mathbf{DL}_{Prog}$:	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 144 **Further information:** "Tool compensation with compensation tables", Page 329

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 326
- Within machining cycles

  Further information: User's Manual for Machining Cycles
- For straight lines **LN** with surface normal vectors

  Further information: "Straight line LN", Page 334

In tool radius compensation, the control takes the delta values from the tool table and the NC program into account.

Active tool radius = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$

R:	Tool radius **R** from the tool table **Further information:** User's Manual for Setup and Program Run
$\mathbf{DR}_{TAB}$:	Tool radius delta value **DR** from the tool table **Further information:** User's Manual for Setup and Program Run
$\mathbf{DR}_{Prog}$:	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 144 **Further information:** "Tool compensation with compensation tables", Page 329

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 487

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 639
- The machine manufacturer uses the optional machine parameter **progToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

  Further information: "Tool call", Page 144

  Further information: User's Manual for Setup and Program Run

11.2 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 141

Requirements

- Defined tool data in the tool management
  Further information: User's Manual for Setup and Program Run

Description of function

The control takes the active tool radius into account during tool radius compensation. The active tool radius results from the tool radius R and the delta values **DR** from the tool management and the **NC program**.

Active tool radius = $\mathbf{R} + \mathbf{DR}_{\mathrm{TAB}} + \mathbf{DR}_{\mathrm{Prog}}$

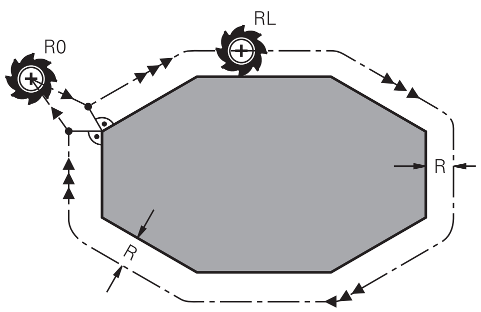
Further information: "Tool compensation for tool length and tool radius", Page 324

Paraxial traverses can be compensated as follows:

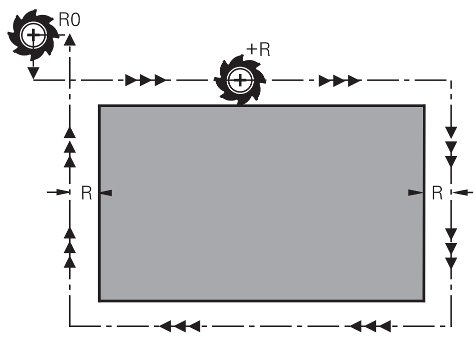
- **R+**: lengthens a paraxial traverse by the amount of the tool radius
- **R-**: shortens a paraxial traverse by the amount of the tool radius

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

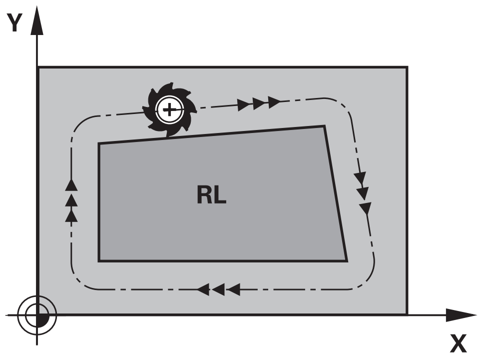


Radius-compensated traverse with path functions

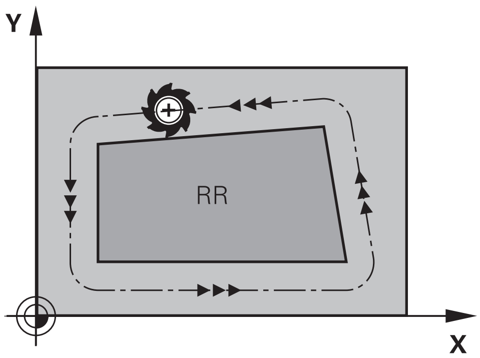


Radius-compensated traverse with paraxial movements

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 implemention 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: User's Manual for Setup and Program Run
- 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.

11.3 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 236

Related topics

- Contents of the compensation tables
  Further information: "Compensation table *.tco", Page 704
  Further information: "Compensation table *.wco", Page 706
- Editing compensation tables during program run
  Further information: User's Manual for Setup and Program Run

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 674
- Activating the compensation table in the NC program
  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 331
- As an alternative, activating the compensation table manually for the program run
  Further information: "Activating the compensation tables manually", Page 330
- Activating a compensation value
  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 332

The compensation table values can be edited within the NC program.

Further information: "Accessing table values ", Page 685

The values in the compensation tables can be edited even while the program is running.

Further information: User's Manual for Setup and Program Run

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 249

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 144

If a shift with the ***.tco** compensation table is active, the control displays it on the **Tool** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

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 244

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: User's Manual for Setup and Program Run

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**.

11.3.1 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 332
- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 704

  Further information: "Compensation table *.wco", Page 706

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

11.3.2 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 331
- Contents of the compensation tables
  Further information: "Compensation table *.tco", Page 704
  Further information: "Compensation table *.wco", Page 706

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

11.4 3D tool compensation (#9 / #4-01-1)

11.4.1 Fundamentals

The control allows 3D tool compensation in CAM-generated NC programs with surface-normal vectors.

Further information: "Straight line LN", Page 334

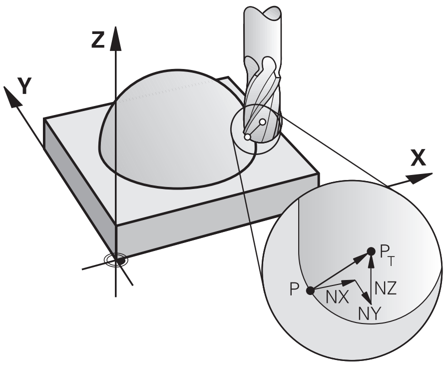
The control displaces the tool in the direction of the surface normals by the total of the delta values from tool management, tool call and compensation tables.

Further information: "Tools for 3D tool compensation", Page 336

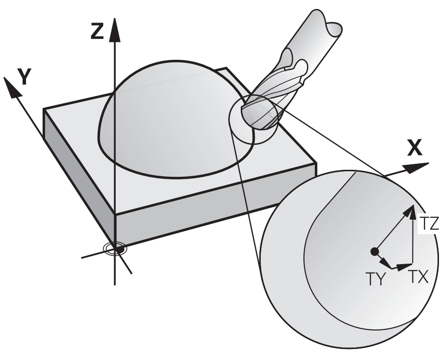
3D tool compensation can be used e. g. in the cases below:

- 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:



> For an optional tool angle of inclination, the NC blocks must include an additional tool vector with the components TX, TY and TZ.



> Note the differences between face milling and peripheral milling.
>
> **Further information:** "3D tool compensation during face milling (#9 / #4-01-1)", Page 337
>
> **Further information:** "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 344

11.4.2 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 333

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 422

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 162

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 326 Optional syntax element
F, **FMAX**, **FZ**, **FU** or **F AUTO**	Feed rate **Further information:** User's Manual for Setup and Program Run 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.

$$\sqrt{NX^2 + NY^2 + NZ^2} = 1$$

11.4.3 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 324
- Compensation in tool call

  Further information: "Tool call by TOOL CALL", Page 144
- Compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 329

Description of function

The tool shapes can be distinguished by columns **R** and **R2** of the tool management:

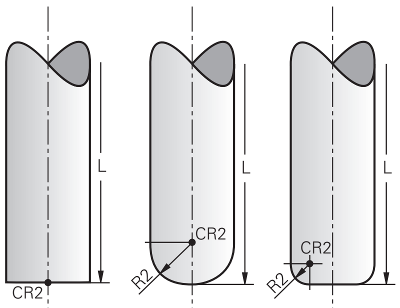
- End mill: **R2** = 0
- Toroid cutter: **R2** > 0
- Ball-nose cutter: **R2** = **R**

Further information: User's Manual for Setup and Program Run

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 141

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 475

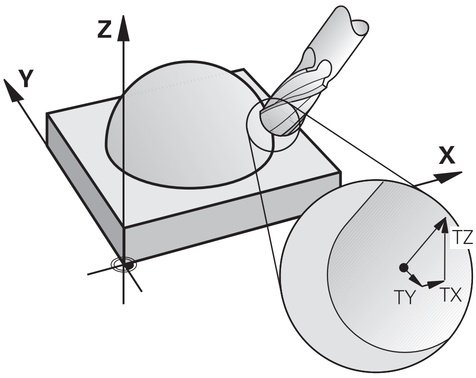
  Use the simulation to ensure that no contours are damaged by the tool oversize.
- The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.If the axis position does not change, you can nevertheless program more than four axes.

11.4.4 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 334
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 460

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315

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

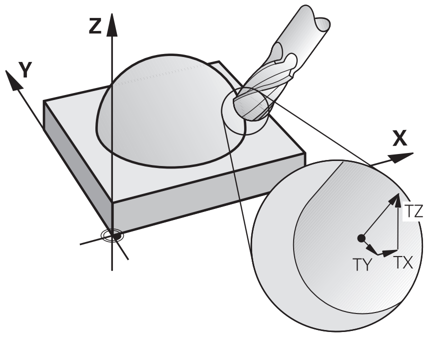
NOTICE

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.

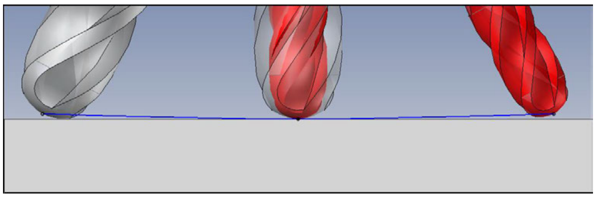


- If a tool orientation **T** has been defined in the **LN** block and **M128** (or **FUNCTION TCPM**) is active at the same time, then the control will position the rotary axes automatically in such a way that the tool can reach the specified tool orientation. If you have not activated **M128** (or **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 347

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:

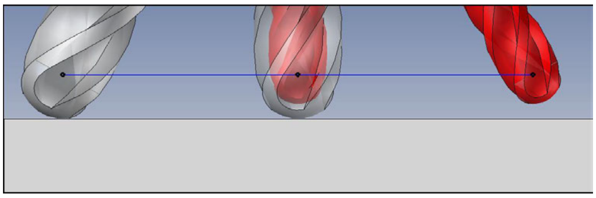
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors

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

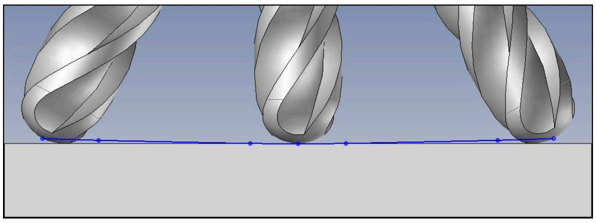
Suggested solution:

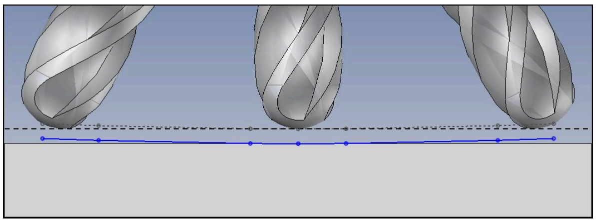
- Tool measurement on tool tip
- TCPM function **REFPNT CNT-CNT**
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DR2** the difference between the nominal value and actual value

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	-0.1	-0.1

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:

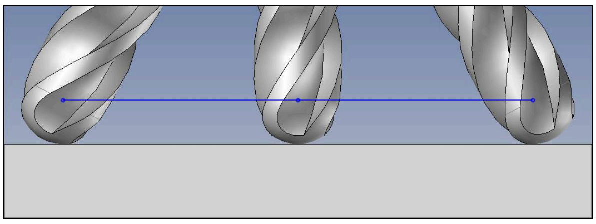
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors and tool vectors

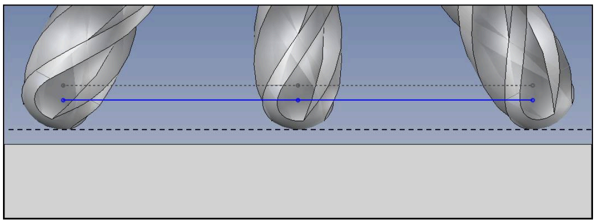
Proposed solution:

- 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**

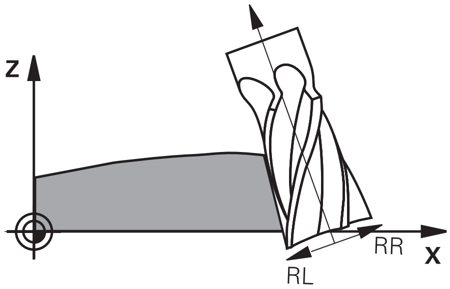
	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	+0	+0
TOOL CALL			+0.2	+0.2	+0.2

11.4.5 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 334
- 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 460

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315
- NC program with tool radius compensation **RL** or **RR**

  Further information: "Tool radius compensation", Page 326

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!

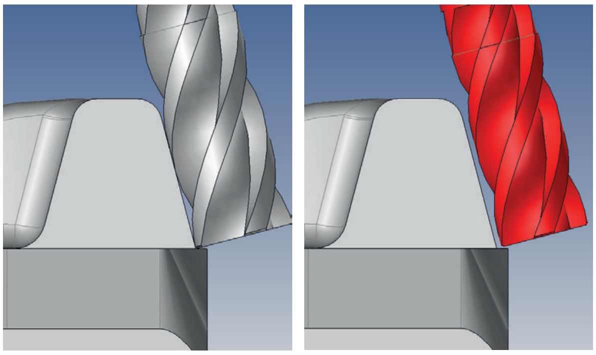
- ▶ Program a safe tool position before the tilting movement, if necessary.
- ▶ Carefully test the NC program or program section in the **Single Block** mode

- 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 347

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

11.4.6 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 333
- Tools for 3D compensation

  Further information: "Tools for 3D tool compensation", Page 336

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 422

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

12

Files

12.1 File management

12.1.1 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
- Connecting network drives

Further information: User's Manual for Setup and Program Run

Description of function

Icons and buttons

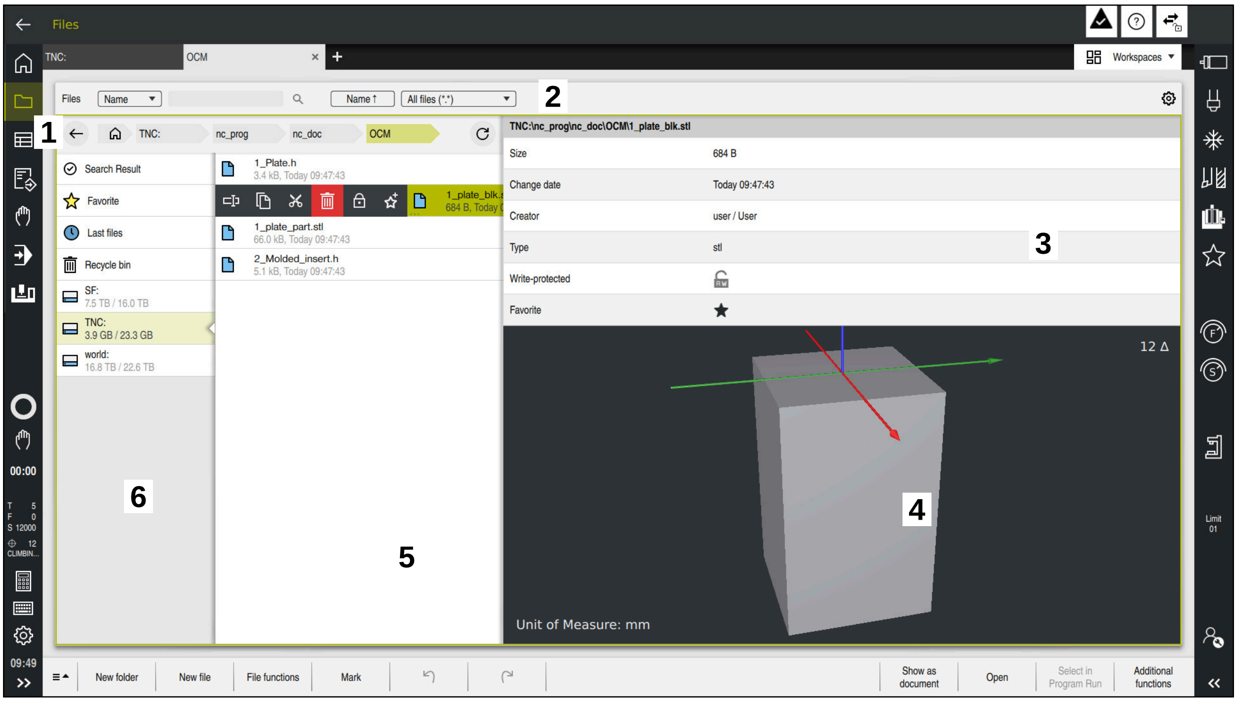
The file management contains the following icons and buttons:

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

Icon, button or shortcut	Meaning
New file	Create new file You create a new table in the **Tables** operating mode. **Further information:** "The Tables operating mode", Page 672
File functions	The control opens the context menu. **Further information:** "Context menu", Page 618 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**
 **CTRL + Y**	**Redo**
Show as document	The control opens the file in the **Document** workspace. **Further information:** "The Document workspace", Page 363
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 365
	■ **Mount network share**
	Further information: User's Manual for Setup and Program Run
	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 353
- Sorting

  Further information: "Sorting in the title bar", Page 353
- Filtering

  Further information: "Filtering in the title bar", Page 353
- Settings

  Further information: "Settings in the title bar", Page 353

3 Information area

Further information: "Information area", Page 354

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 618

6 Navigation column

Further information: "Navigation column", Page 354

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 353

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 355

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 618
- 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 106
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 694
DEP	Automatically generated table with data that depend on the NC program (e.g., the tool usage file) **Further information:** User's Manual for Setup and Program Run
P	Table for pallet-oriented machining **Further information:** "The Job list workspace", Page 654
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) **Further information:** "Point table *.pnt", Page 693
PR	Table with workpiece presets **Further information:** User's Manual for Setup and Program Run
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 690 **Further information:** "Cutting data calculator", Page 625
TCH	Table with the assignment of the tool magazine **Further information:** User's Manual for Setup and Program Run
T	Table with tools for all technologies **Further information:** User's Manual for Setup and Program Run
TP	Table with touch probes (#17 / #1-05-1) **Further information:** User's Manual for Setup and Program Run
TNCDRW	Contour description as a 2D drawing **Further information:** "Graphical programming", Page 555
M3D	Format for tool carriers or collision objects (#40 / #5-03-1), for example **Further information:** "Options for fixture files", Page 382

File type	Application
TNCBCK	File for data backup and restoration **Further information:** User's Manual for Setup and Program Run
EXP	Configuration file for saving and importing configurations of the control interface **Further information:** User's Manual for Setup and Program Run

The control opens these file types with an internal application or with a HEROS tool.

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:** User's Manual for Setup and Program Run
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 641
DXF	2D CAD files
IGS/IGES STP/STEP	3D CAD files **Further information:** User's Manual for Setup and Program Run
CHM	Help files in compiled or compressed format
CFG	Configuration files of the control **Further information:** "Options for fixture files", Page 382 **Further information:** User's Manual for Setup and Program Run
CFT	3D data of a parameterizable tool-carrier template **Further information:** User's Manual for Setup and Program Run
CFX	3D data of a geometrically determined tool carrier **Further information:** User's Manual for Setup and Program Run
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 36
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:** User's Manual for Setup and Program Run
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:** User's Manual for Setup and Program Run

File type	Application
OGG	Container file format for the OGA, OGV, and OGX media types
ZIP	Container file format that collects multiple compressed files.

The control opens some of these file types with the HEROS tools.

Further information: User's Manual for Setup and Program Run

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 672
- 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 532
- 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).
- 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.

12.1.2 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	✓	✓	✓

12.1.3 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 361
- **Editor**

  Further information: "Quick selection new file workspace", Page 362

Further information: "Icons on the control's user interface", Page 74

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 674
- **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 106
New graphical programming	**Contour** **Further information:** "Graphical programming", Page 555
New text file	■ **Text file** with a ***.txt** extension ■ **Format file** with an ***.a** extension **Further information:** "The Text editor workspace", Page 365
New job	**Job list** **Further information:** "The Job list workspace", Page 654

12.1.4 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 356
- **Show as document** button in the **Files** operating mode
  Further information: "Icons and buttons", Page 350

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 59

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 356

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 364
	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.

12.1.5 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 356
- Displaying text files in the **Document** workspace

  Further information: "The Document workspace", Page 363

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 502

Further information: "File types", Page 356

> 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
	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.

12.1.6 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: User's Manual for Setup and Program Run

Adapting a file

Prepare a backup of the original file before adapting

To adapt the format and the content of an iTNC 530 file:

- Select the **Files** operating mode
- Select the desired file
- Additional functions: Select **Additional functions**
- > The control displays a selection menu.
- Select **Update TAB / PGM**
- > The control adapts the file format and content.

> The control saves the changes and overwrites the original file.

- Check the content after adapting

Notes

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.

12.1.7 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

  > The control does not support USB devices with other file systems, such as NTFS.

- A ready data interface

  Further information: User's Manual for Setup and Program Run

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.

- ▶ 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: User's Manual for Setup and Program Run
- 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: User's Manual for Setup and Program Run

12.2 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 356

The control opens the file in the HEROS tool last used for this file type. If you have never opened a file of a certain file type and multiple HEROS tools are available, the control will interrupt program run and open the **Application?** window. In the **Application?** window, you can select the HEROS tool the control should use to open the file. The control saves this selection.

Multiple HEROS tools are available for opening the following file types:

- CFG
- SVG
- BMP
- GIF
- JPG/JPEG
- PNG

> In order to avoid program run interruptions or having to select an alternative HEROS tool, open a file of the corresponding file type once in the file manager. If the files of a certain file type can be opened in multiple HEROS tools, you can use the file manager to select the HEROS tool to be used for opening files of this file type.
>
> **Further information:** "File management", Page 350

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.

13

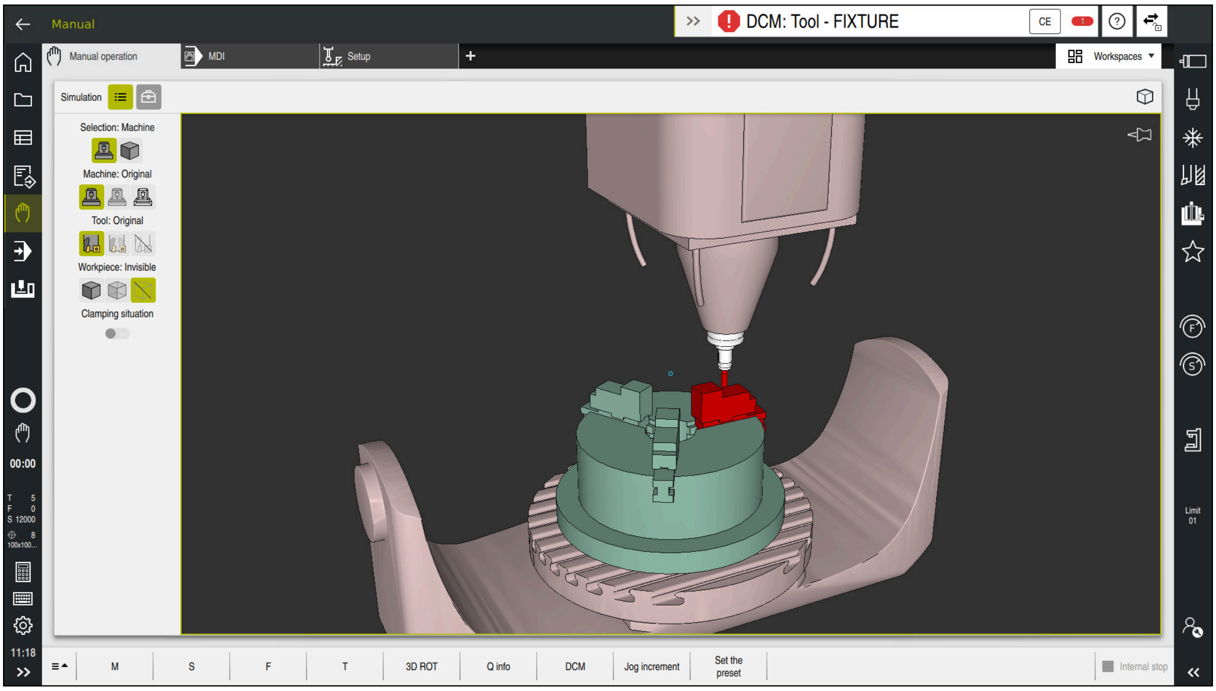
Collision Monitoring

13.1 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 381
- Extended tests in the simulation

  Further information: "Advanced checks in the simulation", Page 388
- Fundamentals of tool carrier management

  Further information: User's Manual for Setup and Program Run
- 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 386

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 381
- Tools with a positive radius **R** and length **L**.

  Further information: User's Manual for Setup and Program Run
- The values in the tool management equal the actual tool dimensions

  Further information: User's Manual for Setup and Program Run

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.

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 388

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: User's Manual for Setup and Program Run

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 379

In the **Editor** operating mode, an NC program can be collision-monitored even prior to execution. In case of collision, the control stops the simulation and displays an error message naming the two objects causing collision.

HEIDENHAIN recommends the use of Dynamic Collision Monitoring (DCM) in the **Editor** operating mode only in addition to DCM in the **Manual** and **Program Run** operating modes.

> The enhanced collision monitoring shows collisions between the workpiece and tools or tool holders.
>
> **Further information:** "Advanced checks in the simulation", Page 388

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 632

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 simulation

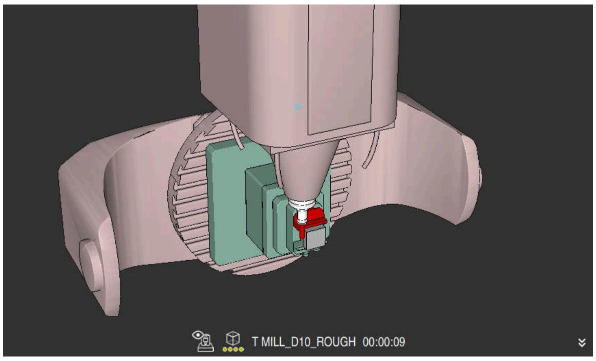
Dynamic Collision Monitoring (DCM) can be activated for the simulation only in the **Editor** operating mode.

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.

> The control displays the status of Dynamic Collision Monitoring (DCM) in the **Simulation** workspace.
>
> **Further information:** "Icons in the Simulation workspace", Page 631

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.

13.1.1 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 386

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

13.2 Fixture management

13.2.1 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 374
- Integrating an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 137

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

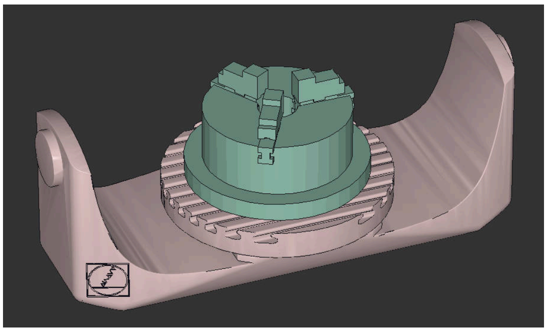
To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading it into the control

  Further information: "Options for fixture files", Page 382
- Fixture placement
  - The **Set up fixtures** function in the **Setup** (#140 / #5-03-2) application

    Further information: User's Manual for Setup and Program Run
  - 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 385



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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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.

13.2.2 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 381

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.

13.2.3 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 374
- Loading and removing the fixture

  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 385

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 374
- Fixture is integrated in the NC program

  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 385

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.

Further information: User's Manual for Setup and Program Run

13.3 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 374

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**

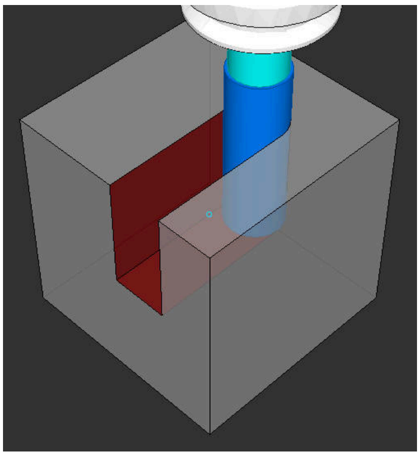
  The control displays a warning in case of collisions between the tool carrier or tool shank and the workpiece.
- **Fixture collision**

  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 632



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 132

13.4 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 470
- Liftoff in the tool axis with **M140**

  Further information: "Retracting in the tool axis with M140", Page 467

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 249

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 374
- 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.

14

Control Functions

14.1 Adaptive feed control (AFC) (#45 / #2-31-1)

14.1.1 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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run
- Define settings for AFC for each tool in the tool management

  Further information: User's Manual for Setup and Program Run
- Define AFC in the NC program

  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 395
- 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 397
- Prior to automatic control, determine the reference spindle power with a teach-in cut

  Further information: User's Manual for Setup and Program Run

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: User's Manual for Setup and Program Run

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run
- 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: User's Manual for Setup and Program Run
- ***.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: User's Manual for Setup and Program Run
- ***.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.

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: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run
  - 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.

  Further information: User's Manual for Setup and Program Run
- 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 352

14.1.2 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: User's Manual for Setup and Program Run
- Desired control setting defined for all tools

  Further information: User's Manual for Setup and Program Run
- **AFC** toggle switch active

  Further information: "The AFC toggle switch in the Program Run operating mode", Page 397

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 395

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: User's Manual for Setup and Program Run

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.

14.2 Functions for controlling program run

14.2.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION S-PULSE	Program pulsing spindle speed	Page 399
FUNCTION DWELL	Program singular dwell time	Page 400
FUNCTION FEED DWELL	Program cyclic dwell time	Page 401

14.2.2 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: User's Manual for Setup and Program Run

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.

14.2.3 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: User's Manual for Machining Cycles
- Program recurring dwell time

  Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 401

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

14.2.4 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 400

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**.

15

Monitoring

15.1 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: User's Manual for Setup and Program Run
- Cycle **238 MEASURE MACHINE STATUS** (#155 / #5-02-1)
  Further information: User's Manual for Machining Cycles
- Color the workpiece as a heat map in the simulation
  Further information: "The Workpiece options column", Page 634

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:

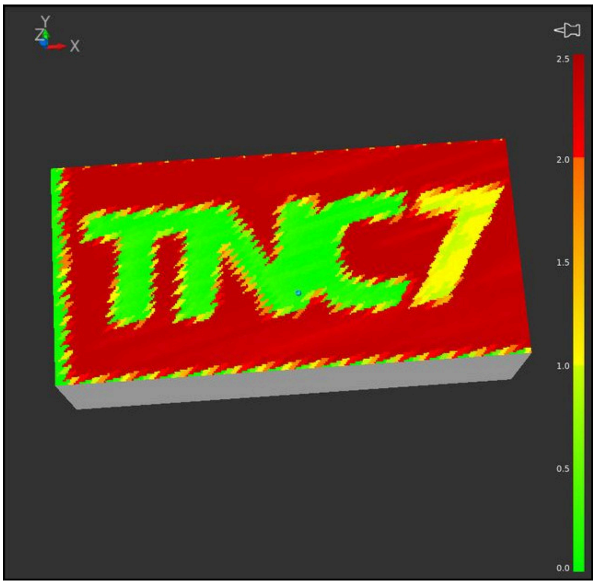
- Green: component works under conditions defined as safe
- Yellow: component works under warning zone conditions
- Red: Overload condition

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)

> 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.

16

Multiple-Axis Machining

16.1 Working with the parallel axes U, V and W

16.1.1 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 102

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 408
- **FUNCTION PARAXMODE**: Select three linear axes for machining

  Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 413

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: User's Manual for Setup and Program Run

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.

16.1.2 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: User's Manual for Setup and Program Run

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 411

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 408

Further information: User's Manual for Setup and Program Run

- 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 104

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**.

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**

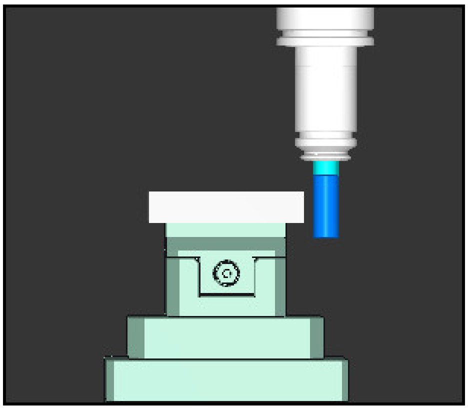
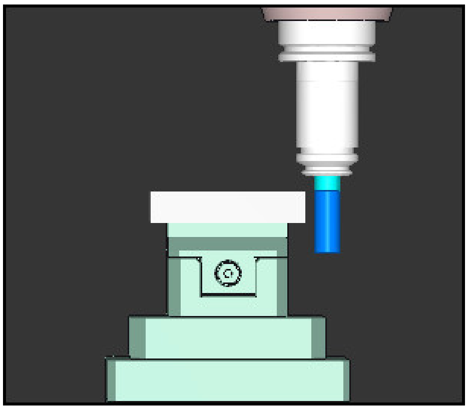
In NC block **13**, the control moves the Z axis to the programmed coordinate **0**.

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: ■ **RFACTL** mode: **Z+0**, **W-30** ■ **ACTL.** mode: **Z-30**, **W-20**	Position-display values: ■ **RFACTL** mode: **Z-10**, **W-30** ■ **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**).

> 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.

16.1.3 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 408

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 414

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: User's Manual for Setup and Program Run

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 162

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: User's Manual for Setup and Program Run

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.

16.1.4 Parallel axes in conjunction with machining cycles

You can also use most machining cycles of the control with parallel axes.

Further information: User's Manual for Machining Cycles

Touch-probe cycles (#17 / #1-05-1) cannot be used in conjunction with parallel axes.

16.1.5 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	

16.2 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

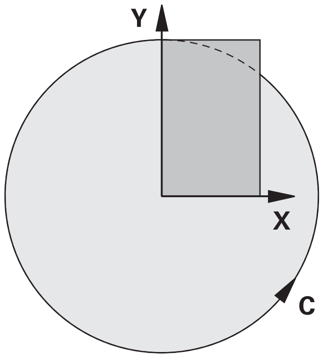
- Machine with at least one rotary axis

  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 408

Description of function

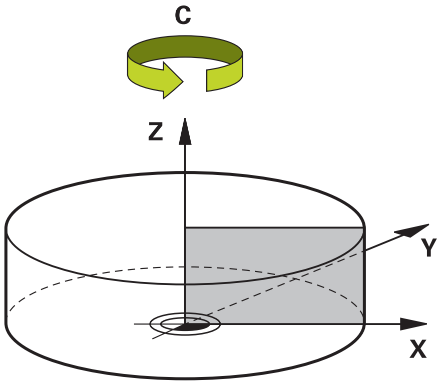


When the polar kinematics are active, the control displays an icon in the **Positions** workspace. This icon covers the icon for the **PARAXCOMP DISPLAY** function.

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

> 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 418 Only when **AXES** is selected
POLE:	Selection of machining in the pole **Further information:** "POLE options", Page 418 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 441
  - 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 432

  Further information: User's Manual for Setup and Program Run

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 **presetToAlignAxis** (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: User's Manual for Setup and Program Run

  - 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 243
  - 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.

16.2.1 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	

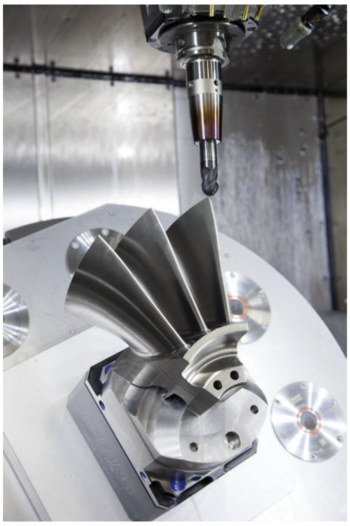
25 L X-20 Y-20 RR	
26 L X+0 Y+20	
27 L X+20 Y-20	
28 L X-20 Y-20	
29 LBL 0	
30 END PGM POLARKIN_SL MM	

16.3 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.

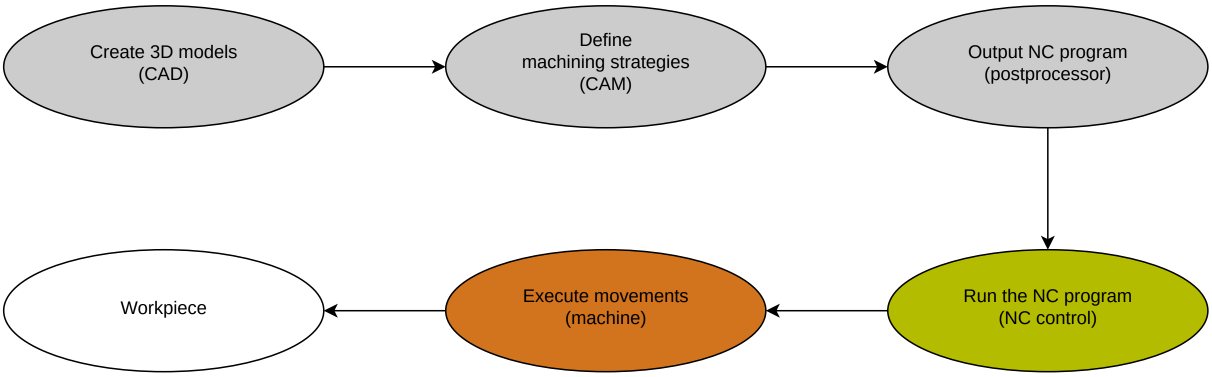


For CAM-generated NC programs to be able to use the full performance potential of the control and to provide you with such options as intervention and correction, certain requirements must be met.

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 427

The process chain specifies the path from a design to the finished workpiece.



Related topics

- Using 3D data directly at the control
  Further information: User's Manual for Setup and Program Run
- Programming graphically
  Further information: "Graphical programming", Page 555

16.3.1 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)

> 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**

> 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.

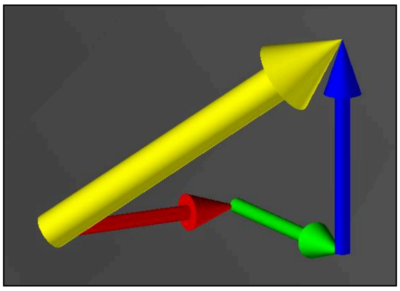
Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315

Further information: "Presets on the tool", Page 141

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.

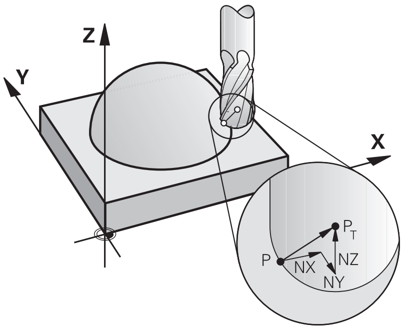
> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1)

> 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.

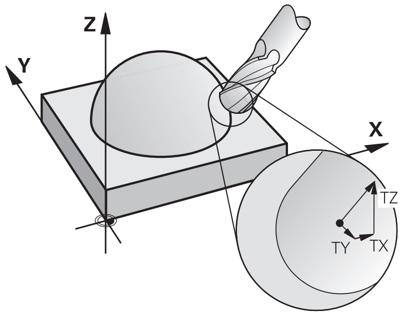
Examples

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

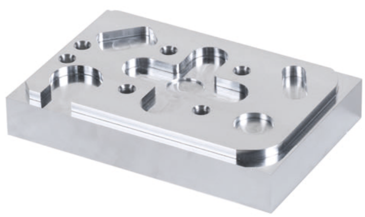
Example

`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

16.3.2 Types of machining according to number of axes

3-axis machining



If only the linear axes **X**, **Y** and **Z** are required for machining a workpiece, 3-axis machining takes place.

3+2-axis machining

If tilting of the working plane is required for machining a workpiece, 3+2-axis machining takes place.

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#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.

> 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.

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (#8 / #1-01-1)
> - Advanced Functions Set 2 (#9 / #4-01-1)

16.3.3 Process steps

CAD

Application

Using CAD systems, designers create the 3D models of the required workpieces. Incorrect CAD data has a negative impact on the entire process chain, including the quality of the workpiece.

Notes

- In 3D models, avoid open or overlapping faces and unnecessary points. If possible, use the check functions of the CAD system.
- Design or save the 3D models based on the center of tolerance and not the nominal dimensions.

> Support manufacturing with additional files:
> - Provide 3D models in STL format. The control-internal simulation can use the CAD data as blank and finished parts, for example. Additional models of tool and workholding equipment are 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

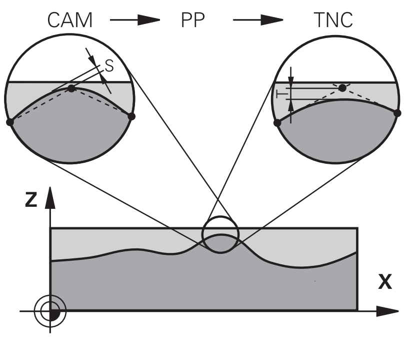
Using machining strategies within the CAM systems, CAM programmers create machine-independent and control-independent NC programs based on the CAD data.

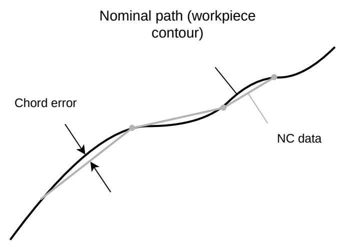
With the aid of the postprocessor, the NC programs are ultimately output specific to machine and control.

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

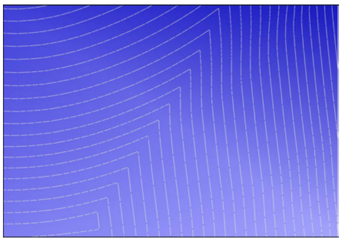
- When finishing with the aim of a high surface quality, the values must allow smoothing of the contour.

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

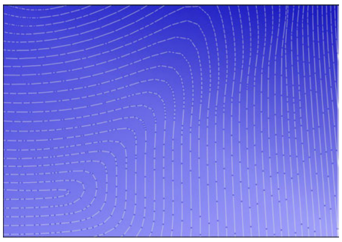
Further information: User's Manual for Machining Cycles

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 269
- 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 315
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 432

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 141

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 480
- 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 221
- Support correction options by outputting contours with tool radius correction.

  Further information: User's Manual for Setup and Program Run
- Use structure items to enable fast navigation within the NC programs.

  Further information: "Structuring of NC programs", Page 610
- Use comments to communicate important information about the NC program.

  Further information: "Adding comments", Page 608

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 629
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 432
- Make sure that the required tools are available and that the remaining service life is sufficient.

  Further information: User's Manual for Setup and Program Run
- If necessary, change the values in Cycle **32 TOLERANCE** depending on the chord error and the dynamic response of the machine.

  Further information: User's Manual for Machining Cycles

  > 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 423
- Software options enable further optimizations.

  Further information: "Functions and function packages", Page 434
  Further information: "Software options", Page 47

Notes on software limit switches for modulo axes

> The following information on software limit switches for modulo axes also applies to traversing limits.
> **Further information:** User's Manual for Setup and Program Run

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

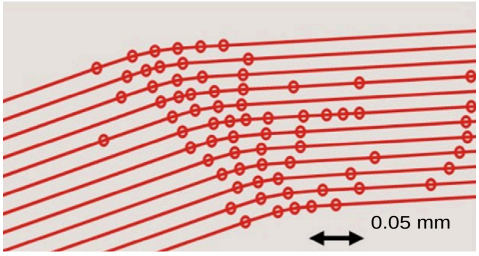
Modulo axis
Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

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.

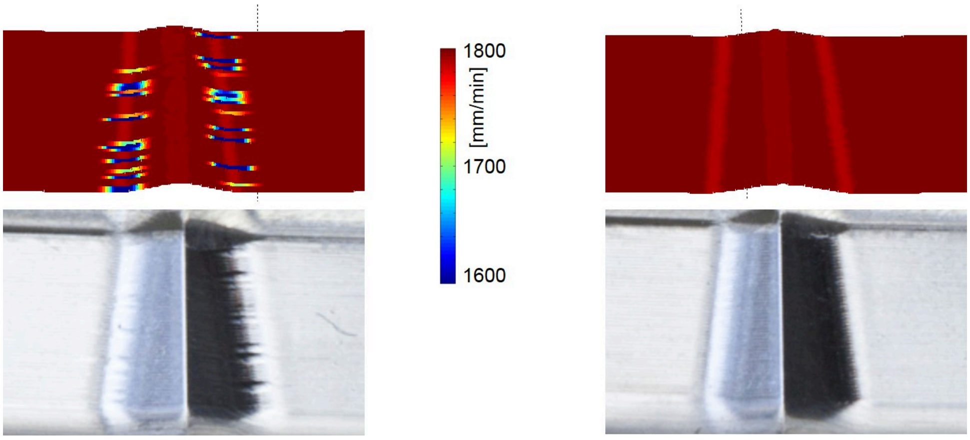
Modulo counting method
The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

16.3.4 Functions and function packages

ADP motion control



Distribution of points



Comparison without and with ADP

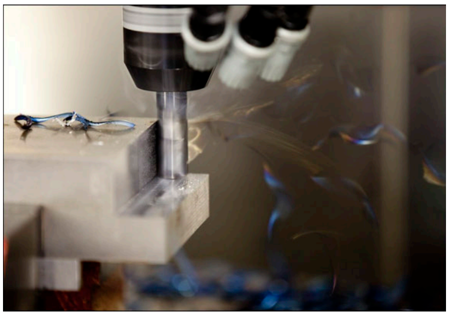
CAM-generated NC programs with an insufficient resolution and variable point density in adjacent paths can lead to feed rate fluctuations and errors on the workpiece surface.

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:

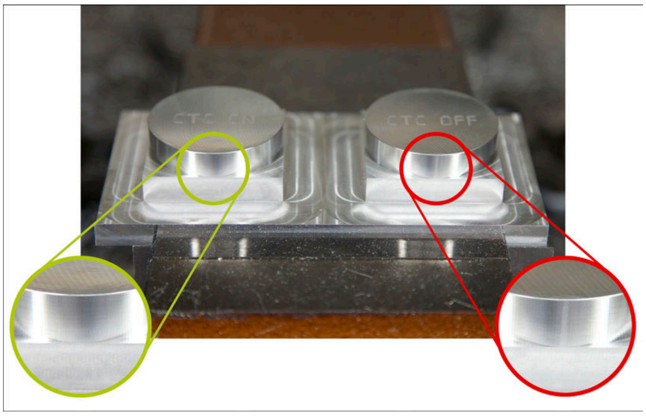
- Active Chatter Control (ACC (#45 / #2-31-1))
- Adaptive Feed Control (AFC (#45 / #2-31-1))
- Trochoidal milling cycles (#167 / #1-02-1)

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.**

17

Miscellaneous Functions

17.1 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: User's Manual for Setup and Program Run

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.

17.1.1 Programming the STOP function

To program the **STOP** function:

STOP

- ▶ Select **STOP**
- \> The control creates a new NC block with the **STOP** function.

17.2 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	□ ■	See the User's Manual for Machining Cycles

Function	Effect	Further information
M91 Traverse in the machine coordinate system **M-CS**	□	Page 441
M92 Traverse in the **M92** coordinate system	□	Page 442
M94 Reduce the display for rotary axes to under 360°	□	Page 444
M97 Machine small contour steps	■	Page 446
M98 Machine open contours completely	■	Page 448
M99 Call a cycle once per block	■	See the User's Manual for Machining Cycles
M101 Automatically insert a replacement tool	□	Page 473
M102 Reset **M101**	■	
M103 Reduce feed rate for infeed movements	□	Page 449
M107 Permit positive tool oversizes	□	Page 475
M108 Check the radius of the replacement tool Reset **M107**	■	Page 477
M109 Adapt feed rate for circular paths	□	Page 450
M110 Reduce feed rate for inner radii	□	
M111 Reset **M109** and **M110**	■	
M116 Interpret feed rate for rotary axes as mm/min	□	Page 452
M117 Reset **M116**	■	
M118 Activate handwheel superimpositioning	□	Page 453
M120 Pre-calculate the radius-compensated contour (look ahead)	□	Page 455
M126 Shorter-path traverse of rotary axes	□	Page 459
M127 Reset **M126**	■	

Function	Effect	Further information
M128 Automatically compensate for tool inclination (TCPM)	□	Page 460
M129 Reset **M128**	■	
M130 Traverse in the non-tilted input coordinate system **I-CS**	□	Page 443
M136 Interpret feed rate as mm/rev	□	Page 465
M137 Reset **M136**	■	
M138 Take rotary axes into account during machining operations	□	Page 466
M140 Retract in the tool axis	□	Page 467
M141 Suppress touch probe monitoring	□	Page 478
M143 Rescind basic rotations	□	Page 469
M144 Factor the tool offset into the calculations	□	Page 469
M145 Reset **M144**	■	
M148 Automatically lift off upon an NC stop or a power failure	□	Page 470
M149 Reset **M148**	■	
M197 Prevent rounding off of outside corners	■	Page 471

17.3 Miscellaneous functions for coordinate entries

17.3.1 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 238

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 104

> 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: User's Manual for Setup and Program Run
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 104
- 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)**
- 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 632
- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

17.3.2 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 104

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 104

> 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: User's Manual for Setup and Program Run
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 104
- 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 632
- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

17.3.3 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 162

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 248

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.

17.4 Miscellaneous functions for path behavior

17.4.1 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: User's Manual for Setup and Program Run

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 306
- 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°.

17.4.2 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.

> 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)

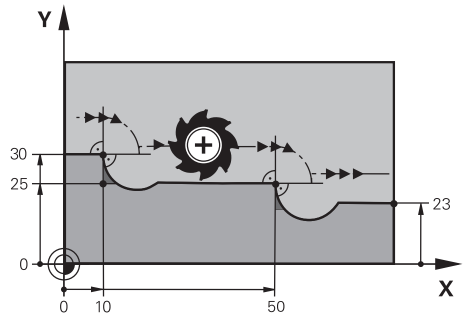
  Further information: "Pre-calculating a radius-compensated contour with M120 (#21 / #4-02-1)", Page 455

Description of function

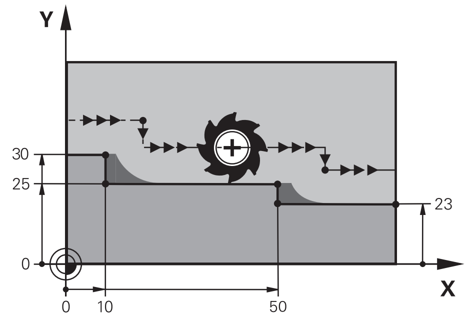
Effect

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.

17.4.3 Machining open contour corners with M98

Application

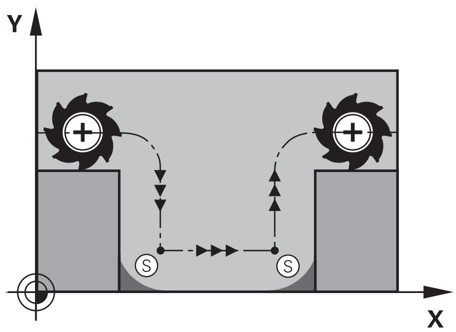
If the tool performs a machining operation on a radius-compensated contour, then residual material remains at the inside corners. With **M98** the control extends the tool path by the tool radius so that the tool completely machines an open contour and removes all residual material.

Description of function

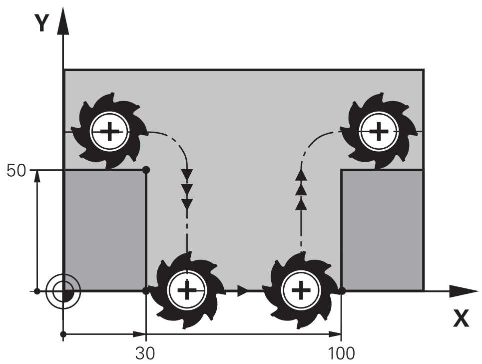
Effect

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.

17.4.4 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**.

17.4.5 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.

17.4.6 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.

17.4.7 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 268
- 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 466
- Without **M128** or **FUNCTION TCPM** (#9 / #4-01-1), **M116** can be in effect for multiple rotary axes at the same time.

17.4.8 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.

> 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.

  Further information: User's Manual for Setup and Program Run
- 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: User's Manual for Setup and Program Run
- Handwheel superimpositioning is also effective in the **MDI** application.

  Further information: User's Manual for Setup and Program Run
- If you want to use **M118** with clamped axes, you must unclamp them first.

17.4.9 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

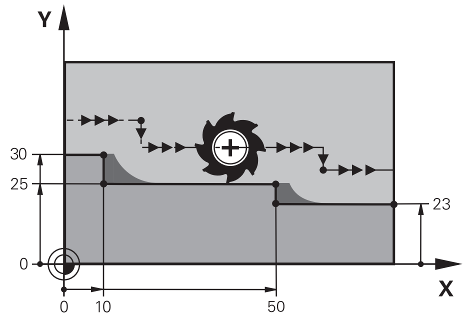
Effect

M120 takes effect at the start of the block and remains active beyond the milling cycles.

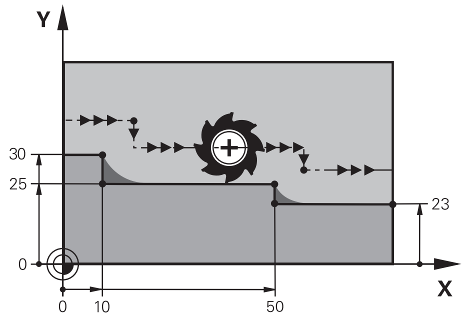
M120 can be reset by the following NC functions:

- **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!

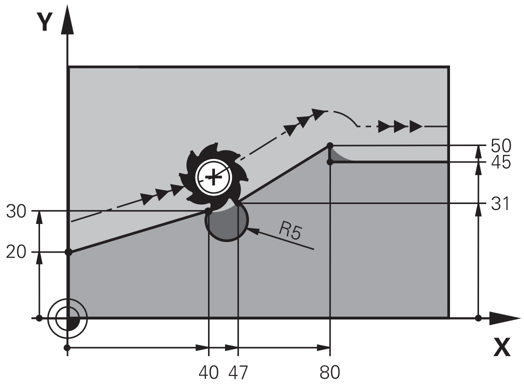
Define as low a number as possible of **LA** NC blocks to be pre-calculated. If the value defined is too large, the control might overlook parts of the contour!

- 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

17.4.10 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 306
- 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°.

17.4.11 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).

> 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 315

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

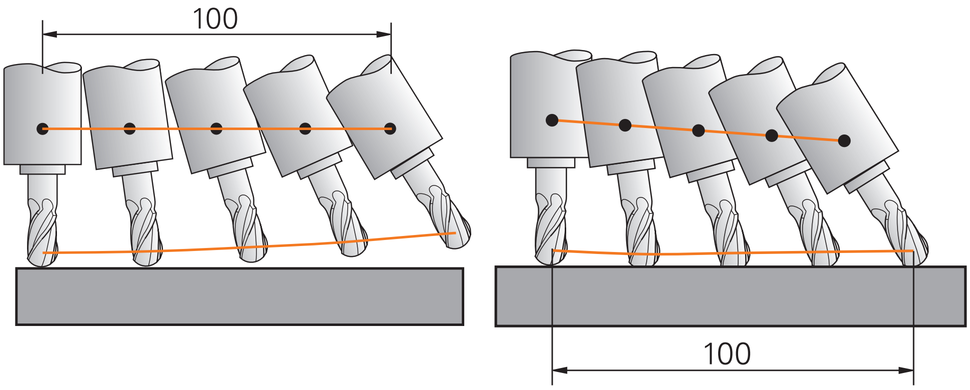
M128 takes effect at the start of the block.

You can reset **M128** with the following functions:

- **M129**
- **FUNCTION RESET TCPM**
- In the **Program Run** operating mode, select a different NC program

> **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 141
- > 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

> 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 344

Further information: "Output with vectors", Page 424

- 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: User's Manual for Setup and Program Run
- **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: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315
- 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 236
- 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: User's Manual for Setup and Program Run

  - 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 243
  - 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 141
- 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 325

  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 374
- 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 315

Definition

Abbreviation	Definition
TCPM (tool center point management)	Maintain the position of the tool location point **Further information:** "Presets on the tool", Page 141

17.4.12 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: User's Manual for Setup and Program Run
- **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.

17.4.13 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 460
- **FUNCTION TCPM** (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 315
- **PLANE** functions (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 269
- 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.

17.4.14 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 249
- 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

17.4.15 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.

Note

The function **M143** is not permitted with mid-program startup.

Further information: User's Manual for Setup and Program Run

17.4.16 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.

> 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 315

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 441
- 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 269
  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 236

17.4.17 Automatically lifting off upon an NC stop or a power failure with M148

Application

With **M148** the control automatically retracts the tool from the workpiece in the following situations:

- Manually triggered NC stop
- NC stop triggered by the software, for example if an error has occurred in the drive system
- Power interruption

> Instead of **M148**, HEIDENHAIN recommends using the more powerful function **FUNCTION LIFTOFF**.

Related topics

- Automatic retraction with **FUNCTION LIFTOFF**
  Further information: "Automatic tool liftoff with FUNCTION LIFTOFF", Page 389

Requirement

- **LIFTOFF** column in the tool management
  You must define the value **Y** in the **LIFTOFF** column of the tool management.
  Further information: User's Manual for Setup and Program Run

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.

17.4.18 Preventing rounding off of outside corners with M197

Application

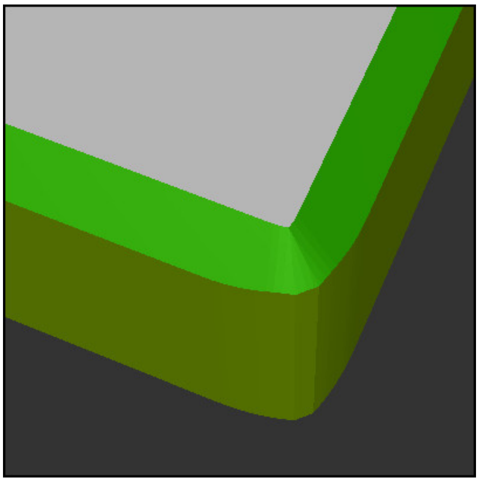
With **M197** the control extends a radius-compensated contour at the corner tangentially and inserts a smaller transition arc. That way you prevent the tool from rounding off the outside corner.

Description of function

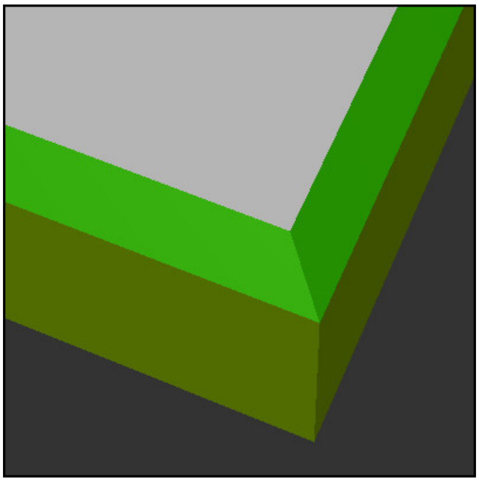
Effect

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

17.5 Miscellaneous functions for tools

17.5.1 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: User's Manual for Setup and Program Run

> 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 477

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: User's Manual for Setup and Program Run
- 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: User's Manual for Setup and Program Run
- 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: User's Manual for Setup and Program Run
- The control cannot perform the automatic tool change at the following points in a program.
  - During a machining cycle
  - If radius compensation with **RR** or **RL** is active
  - Directly after an **APPR** approach function
  - Directly before a **DEP** departure function
  - Directly before and after a chamfer with **CHF** or a rounding with **RND**
  - During a macro
  - During a tool change
  - Directly after the NC functions **TOOL CALL** or **TOOL DEF**
- If the machine manufacturer does not define otherwise, the control moves the tool after the tool change as follows:
  - If the target position in the tool axis is below the current position, the tool axis is positioned last.
  - If the target position in the tool axis is above the current position, the tool axis is positioned first.

Notes on the input value BT

- To calculate a suitable initial value for **BT**, use the following formula:

  $BT = 10 \div t$

  t: average machining time of an NC block in seconds

  Round the result up to an integer value. If the calculated result is greater than 100, use the maximum input value of 100.
- In the optional machine parameter **M101BlockTolerance** (no. 202206) the machine manufacturer defines the standard value for the number of NC blocks by which the automatic tool change may be delayed. This standard value applies if you do not define **BT**.

Definition

Abbreviation	Definition
BT (block tolerance)	Number of NC blocks by which a tool change may be delayed.

17.5.2 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 333

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 423

Requirement

- Software option Advanced Functions Set 2 (#9 / #4-01-1)

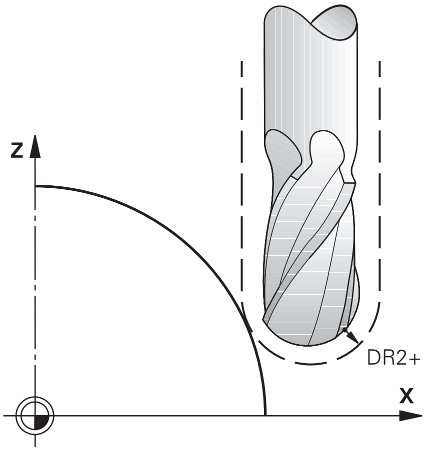
Description of function

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 344
- 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 337

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

17.5.3 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 473

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 475

17.5.4 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: User's Manual for Measuring Cycles for Workpieces and Tools

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

18

Variable Programming

18.1 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 494
Trigonometric functions	Page 496
Circle calculations	Page 498
Jump commands	Page 500
Special functions	Page 501 Page 513
SQL statements	Page 532
String functions	Page 522
Counters	Page 531
Calculations using formulas	Page 517
Function for the definition of complex contours	See the User's Manual for Machining Cycles

18.2 Variables: Q, QL, QR and QS parameters

18.2.1 Basics

Application

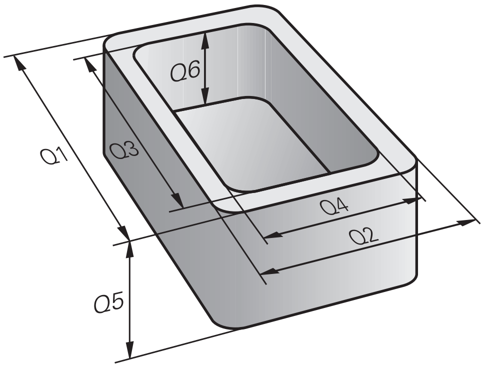
You can use the Q, QL, QR and QS parameters of the control, also referred to as variables, to take measurement results into account dynamically within calculations while machining.

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: User's Manual for Setup and Program Run

Variable types

The control provides the following variables for numerical values:

- Q parameters
  Further information: "Q parameters", Page 482
- QL parameters
  Further information: "QL parameters", Page 482
- QR parameters
  Further information: "QR parameters", Page 482

In addition, the control provides QS parameters for alpha-numeric values (e.g., texts).

Further information: "QS parameters", Page 482

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 ; ! # $ % & ' ( ) + , - . / : < = > ? @ [ ] ^ _ ` *

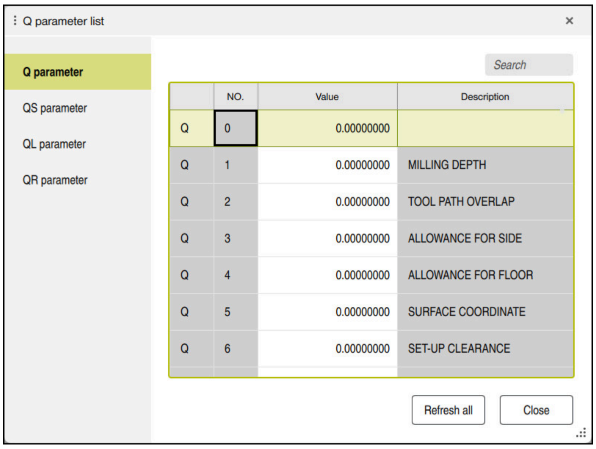
QS parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program.

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.

> 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:** User's Manual for Setup and Program Run
>
> 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 482

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 485

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

> 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 487

- 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 604
- 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 496
- 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: User's Manual for Setup and Program Run

18.2.2 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

> The control will remember the active tool radius even after a restart of the control.

Further information: User's Manual for Setup and Program Run

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 102

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 437

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: User's Manual for Machining Cycles

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

> The control remembers the active tool length even after a restart of the control.

Further information: User's Manual for Setup and Program Run

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.

> 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 602

Further information: User's Manual for Measuring Cycles for Workpieces and Tools

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

> 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

> 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

18.2.3 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 487
- Calculations using formulas

  Further information: "Formulas in the NC program", Page 517

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.

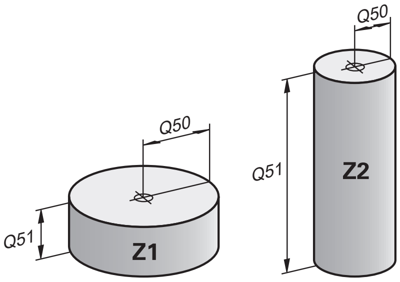
To the right of the equal sign, you can use fixed or variable values. The variables and numerical values in the equations can be entered with an algebraic sign.

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:

Insert NC function

- ▶ 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**).

18.2.4 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

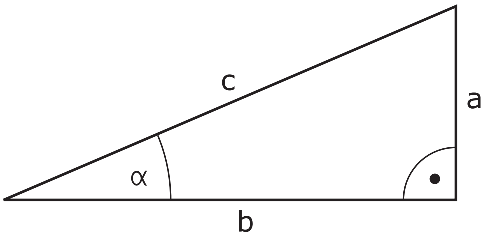
The **Trigonometric functions** folder contains the following functions:

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 trigonometric 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 $a^2 = a*a$)

$c = \sqrt{(a^2+b^2)}$

11 Q50 = ATAN (+25 / +50)	Calculate angle **α**
12 FN 8: Q51 = +25 LEN +50	Calculate side length c

18.2.5 The Circle calculation folder

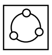
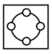
Application

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 102

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.

18.2.6 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 222

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 224

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

18.2.7 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 709
- Error messages in the notification menu

  Further information: User's Manual for Setup and Program Run

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 709

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: User's Manual for Setup and Program Run
- Print to a connected printer
  Further information: User's Manual for Setup and Program Run

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 503

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 505

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 365

Formatting

The formatting of the source file can be defined with the following formatting characters:

> Please note that the input is case-sensitive.

Formatting characters	Meaning
“...“	Identifies the formatting of the contents to be output 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: User's Manual for Setup and Program Run

Example

11 FN 16: F-PRINT TNC:\MASKE - \MASKE1.A / SCREEN:	; Display the output file with **FN 16** on the control screen

> 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: User's Manual for Setup and Program Run

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 113

  Further information: User's Manual for Setup and Program Run

Example

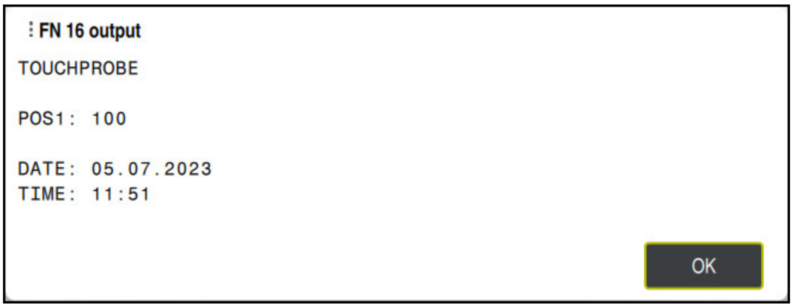
Example of a format file that generates an output file with variable contents:

“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 714
- Read system data using QS parameters
  Further information: "Read system data with SYSSTR", Page 524

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 686

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 503
/	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

18.2.8 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 690
- Access to table values in case of low computing power

  Further information: "Table access with SQL statements", Page 532

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 690
- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 513

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:** User's Manual for Setup and Program Run
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 496

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 690
- Opening a freely definable table
  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 513
- Writing a freely definable table
  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 513

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.

18.2.9 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 522
- Define a single calculation in an NC block

  Further information: "The Basic arithmetic folder", Page 494

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 521
- 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 604

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 521

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 519

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 **PI**	**Use the “pi” constant** π = 3.14159 Example: **Q15 = PI**	
LN **LN**	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG **LOG**	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP **EXP**	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG **NEG**	**Negate** Multiply by –1 Example: **Q2 = NEG Q1**	Function
INT **INT**	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42** i The **INT** function does not round off—it simply truncates the decimal places. Input: **0...999999999**	Function
ABS **ABS**	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC **FRAC**	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN **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 494

Further information: "The Trigonometric functions folder", Page 496

You can also define arithmetic operations for strings.

Further information: "String functions", Page 522

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

18.3 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 482

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 526	
STRING FORMULA	Concatenate contents of QS parameters and assign them to a QS parameter **Further information:** "Concatenation of alphanumeric values", Page 527	**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 527	**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 528	**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 528	**String formula QS**
SYSSTR	Read system data and assign the contents to a QS parameter **Further information:** "Read system data with SYSSTR", Page 524	**String formula QS**

Syntax	NC function	Higher-level NC function
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 528	**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 528	**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 529	**Formula Q/QL/QR**
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 530	■ **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 521
- 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 604

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 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

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 530

18.3.1 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**

18.3.2 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**

18.3.3 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**

18.3.4 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**

18.3.5 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**

18.3.6 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**

18.3.7 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**

18.3.8 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)

> 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**

18.3.9 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

  > 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 525

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.

18.4 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: User's Manual for Setup and Program Run

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 531

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

NOTICE
Caution: Data may be lost!
Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.
▶ Please check prior to machining whether a counter is active.

- The machine manufacturer uses the optional machine parameter **CfgNcCounter** (no. 129100) to define whether you can edit the counter.
- You can engrave the current counter reading with Cycle **225 ENGRAVING**.

  Further information: User's Manual for Machining Cycles

18.4.1 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

18.5 Table access with SQL statements

18.5.1 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 513

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function

In the NC software, table accesses occur through an SQL server. This server is controlled via the available SQL commands. The SQL commands can be defined directly in an NC program.

The server is based on a transaction model. A **transaction** consists of multiple steps that are executed together, thereby ensuring that the table entries are processed in an orderly and well-defined manner.

The SQL commands take effect in the **Program Run** operating mode and the **MDI** application.

Example of transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL EXECUTE** with the instruction **SELECT**
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, or **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** or **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**

> You must conclude all transactions that have been started—even exclusively reading accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

The **result set** contains a subset of a table file. It results from a **SELECT** query performed on the table.

The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.

This query has the same effect as applying a filter to the table, so that only part of the data records become visible. To perform this query, the table file must be read at this point.

The SQL server assigns a **handle** to the **result set**, which enables you to identify the result set for reading or editing data and completing the transaction. The **handle** is the result of the query, which is visible in the NC program. The value 0 indicates an **invalid handle**, i.e. it was not possible to create a **result set** for that query. If no rows are found that satisfy the specified condition, an empty **result set** is created and assigned a valid **handle.**

Overview of SQL commands

The control provides the following SQL commands:

Syntax	Function	Further information
SQL BIND	**SQL BIND** creates or disconnects a binding between table columns and Q or QS parameters	Page 535
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	Page 536
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 539
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	Page 544
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	Page 545
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	Page 547
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	Page 548
SQL INSERT	**SQL INSERT** creates a new table row	Page 550

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.

18.5.2 Binding a variable to a table column with SQL BIND

Application

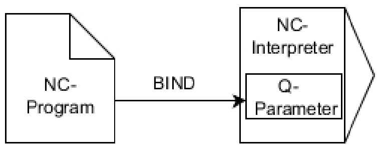
SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

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 539
- 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.

18.5.3 Reading out a table value with SQL SELECT

Application

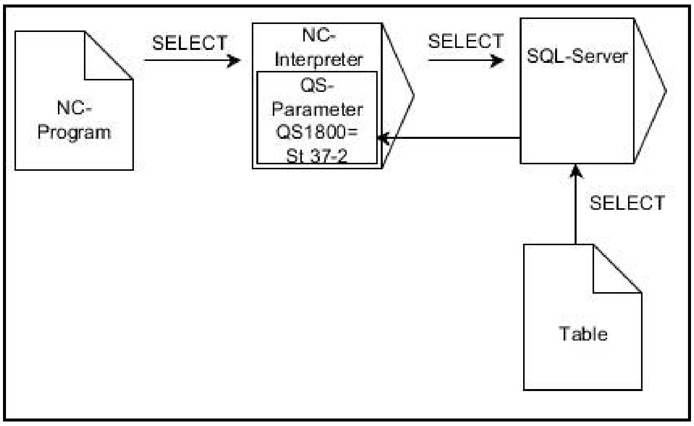
SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

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 527
- 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: User's Manual for Setup and Program Run

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	
* - ...	

18.5.4 Executing SQL statements with SQL EXECUTE

Application

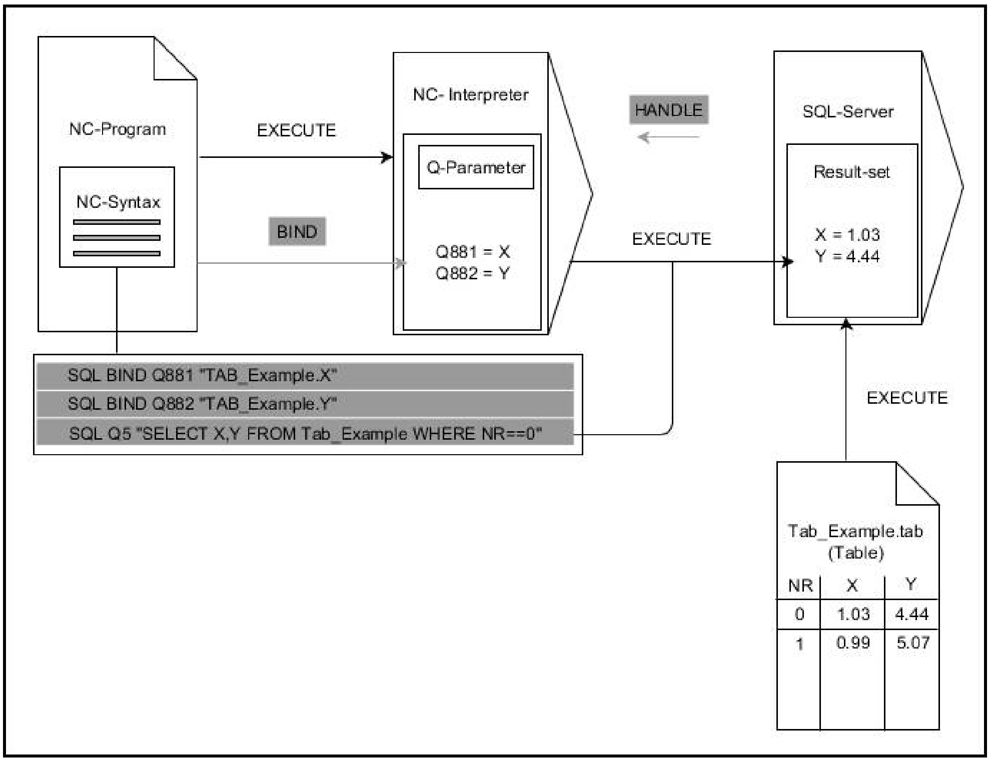
SQL EXECUTE can be used in conjunction with various SQL instructions.

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 527
- 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: User's Manual for Setup and Program Run

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:      ; Create synonym
   \table\NewTab.TAB'"
2  SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z  ; Create table
   FROM 'TNC:\prototype_for_NewTab.tab'"
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
```

18.5.5 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**.

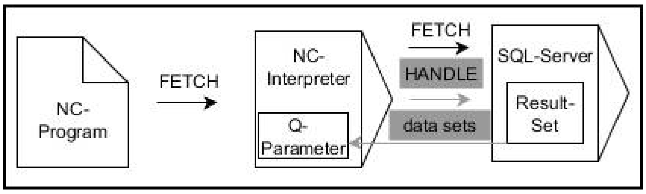
SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

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
```

18.5.6 Discarding changes to a transaction using SQL ROLLBACK

Application

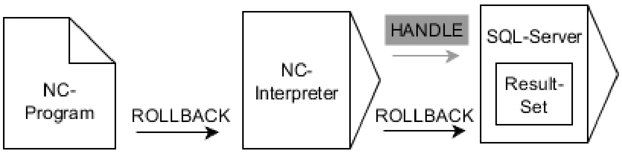
SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

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
```

18.5.7 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.

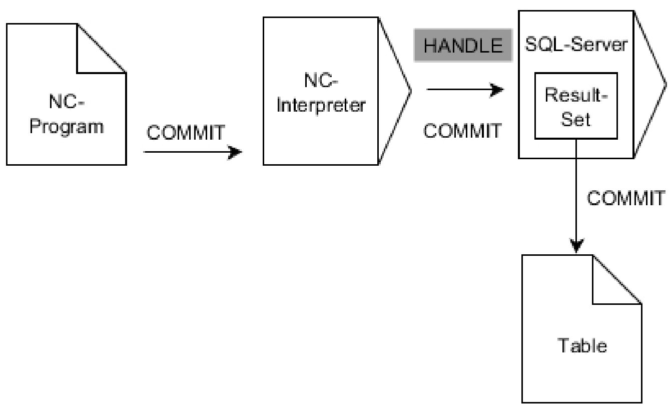
Requirements

- 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
```

18.5.8 Changing the row of a result set with SQL UPDATE

Application

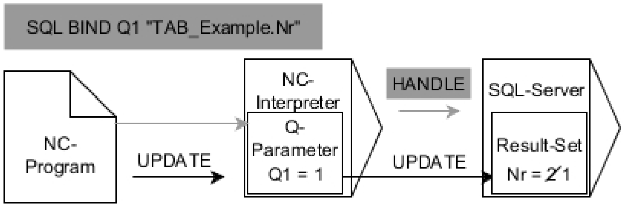
SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing 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
```

18.5.9 Creating a new row in the result set with SQL INSERT

Application

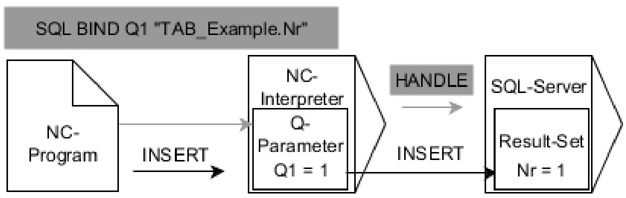
SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

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
```

18.5.10 Example

In the following example, the defined material is read from the table (**WMAT.TAB**) and is stored as a text in a QS parameter. The following example shows a possible application and the necessary program steps.

> You can use the **FN 16** function, for example, in order to reuse QS parameters in your own log files.

Use synonym

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table-\WMAT.TAB'"	; Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	; Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	; Define search
4 SQL FETCH Q1900 HANDLE QL1	; Execute search
5 SQL ROLLBACK Q1900 HANDLE QL1	; Complete transaction
6 SQL BIND QS1800	; Remove parameter binding
7 SQL Q1 "DROP SYNONYM my_table"	; Delete synonym
8 END PGM SQL_READ_WMAT MM	

Step	Explanation
1 Create synonym	Assign a synonym to a path (replace long paths with short names) ▪ The path **TNC:\table\WMAT.TAB** is always placed in single quotes ▪ The selected synonym is **my_table**
2 Bind QS parameters	Bind a QS parameter to a table column ▪ **QS1800** is freely available in NC programs ▪ The synonym replaces the entry of the complete path ▪ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ▪ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) ▪ The synonym defines the table ▪ The **WMAT** entry defines the table column of the read operation ▪ The entries **NR** and **==3** define the table rows of the read operation ▪ Selected table columns and rows define the cells of the read operation

4 Execute search	The control performs the read operation
	▪ **SQL FETCH** copies the values from the **result set** into the bound Q or QS parameter
	  ▪ **0** successful read operation
	  ▪ **1** faulty read operation
	▪ The syntax **HANDLE QL1** is the transaction designated by the parameter **QL1**
	▪ The parameter **Q1900** is a return value for checking whether the data have been read

5 Complete transaction	The transaction is concluded and the used resources are released

Step	Explanation
6 Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7 Delete synonym	The synonym is deleted (release of necessary resources)

> Synonyms are an alternative only to the required absolute paths. Relative path entries are not possible.

The following NC program shows the entry of an absolute path.

0 BEGIN PGM SQL_READ_WMAT_2 MM	
1 SQL BIND QS 1800 "'TNC:\table-\WMAT.TAB'.WMAT"	; Bind QS parameters
2 SQL QL1 "SELECT WMAT FROM 'TNC:-\table\WMAT.TAB' WHERE NR ==3"	; Define search
3 SQL FETCH Q1900 HANDLE QL1	; Execute search
4 SQL ROLLBACK Q1900 HANDLE QL1	; Complete transaction
5 SQL BIND QS 1800	; Remove parameter binding
6 END PGM SQL_READ_WMAT_2 MM	

19

Graphical programming

19.1 Fundamentals

Application

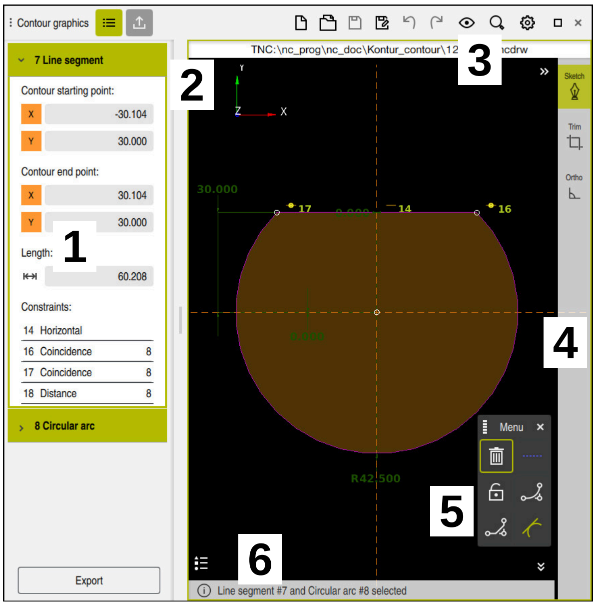
Graphical programming offers an alternative to conventional Klartext programming. You can create 2D sketches by drawing lines and arcs and generate a contour from this in Klartext. In addition, you can import existing contours from an NC program into the **Contour graphics** workspace and edit them graphically.

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 570

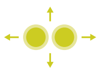
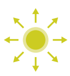
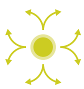
The following elements are available in graphical programming:

- 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 65

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 74

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 562
	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 562

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.

Icon or shortcut	Designation	Meaning
//	**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**. 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.

Icon or shortcut	Designation	Meaning
CTRL + T	**Trim**	If multiple elements overlap, you can use **Trim** mode to shorten elements to the next adjacent element. **Trim** mode remains active until you deactivate it again. If the function is active, the control changes the button to green.
	Ortho	With this function, you can only draw rectangular lines. The control does not allow oblique lines or arcs. If the function is active, the control changes the button to green.
CTRL + A	**Select all**	The **Select All** function allows you to mark all drawn elements at once.

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.

19.1.1 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.

19.1.2 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 562
- 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 560
- 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

19.2 Importing contours into graphical programming

Application

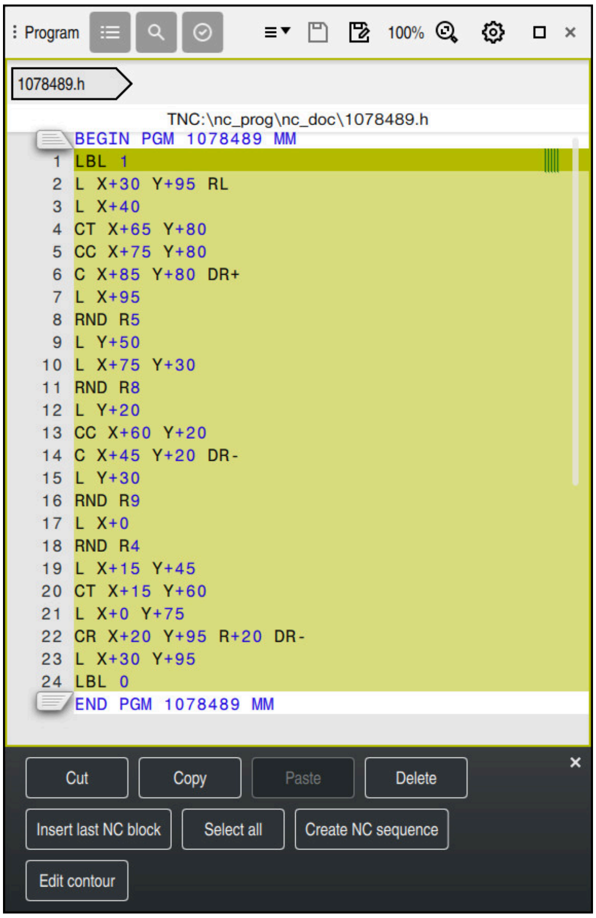
In the **Contour graphics** workspace, you can not only create new contours, but also import contours from existing NC programs and, if necessary, edit them graphically.

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:

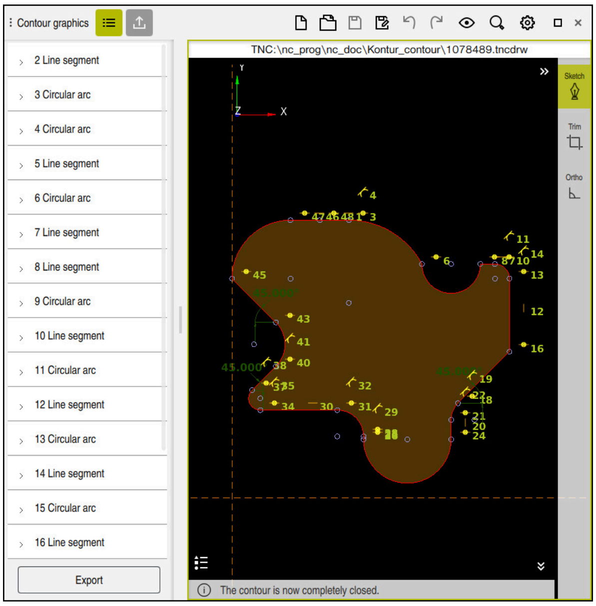
- Circular contour **CT**

  Further information: "Circular path CT", Page 175
- NC blocks with polar coordinates

  Further information: "Polar coordinates", Page 154
- NC blocks with incremental inputs

  Further information: "Incremental entries", Page 157
- Free contour programming **FK**

19.2.1 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.

> 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 65

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 563
- You can edit contours graphically and export them after importing.

  Further information: "First steps in graphical programming", Page 570

  Further information: "Exporting contours from graphical programming", Page 567
- 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**).

19.3 Exporting contours from graphical programming

Application

The **Export** column in the **Contour graphics** workspace allows you to export newly created or graphically edited contours.

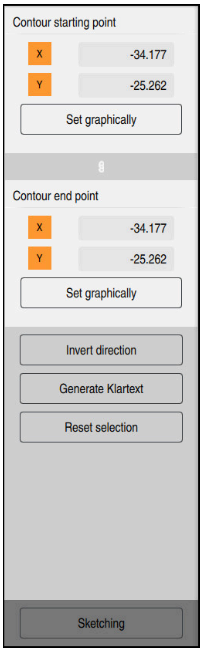
Related topics

- Importing contours

  Further information: "Importing contours into graphical programming", Page 564
- First steps in graphical programming

  Further information: "First steps in graphical programming", Page 570

Description of function



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

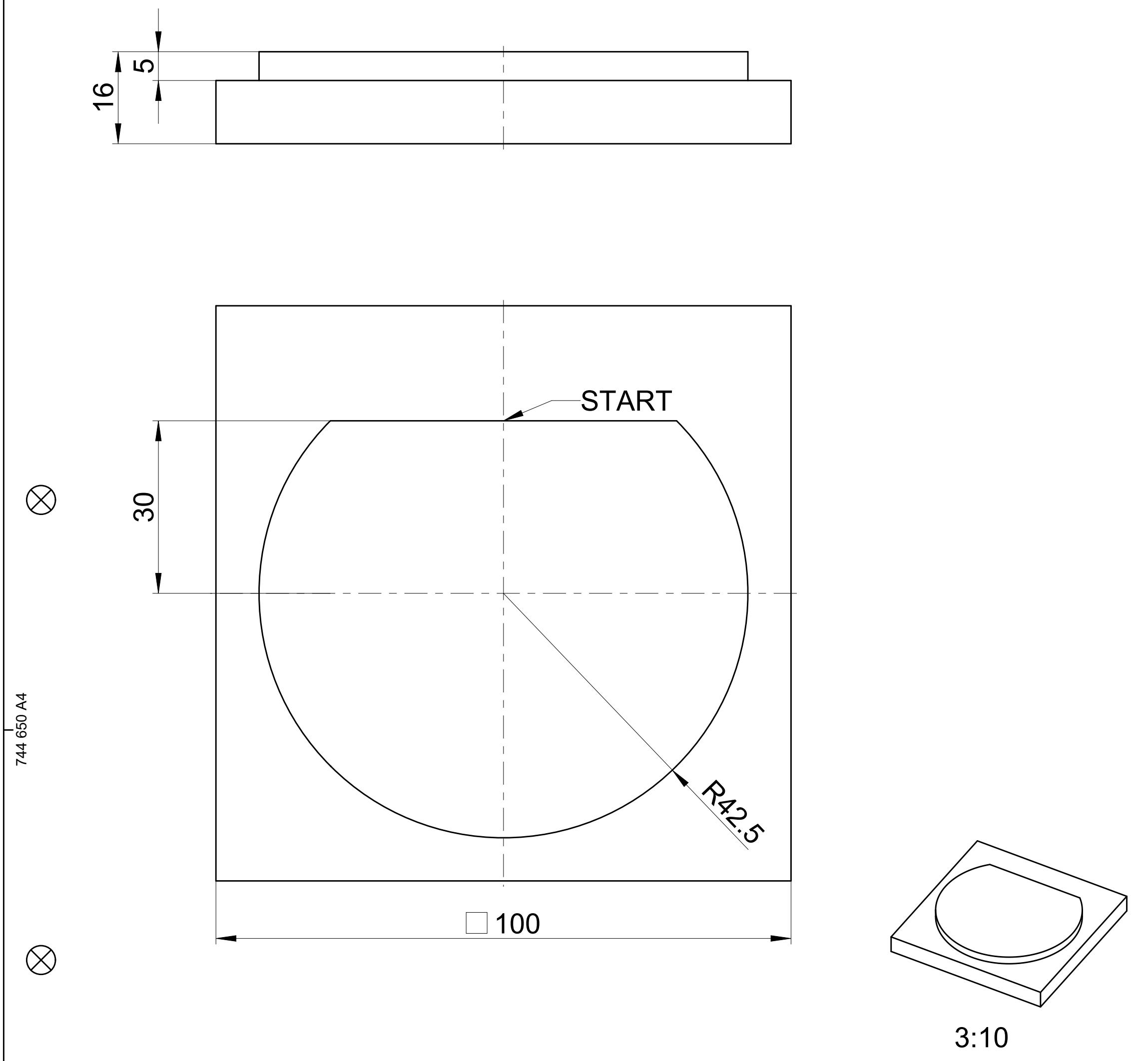
Icon or button	Meaning
Set graphically	Graphically set the Contour starting point or Contour end point
	**Closed contour** In a closed contour, the starting and end point coincide. When you select the starting point, the control will set the end point automatically.
	**Open contour** In an open contour, 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 562 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.

19.4 First steps in graphical programming

19.4.1 Example task D1226664



ID number

Text:

Change No. C000941-05
Phase: Nicht-Serie

Original drawing

	Scale	Format
RoHS	1:1	A4

Maße in mm / Dimensions in mm

Platte

Plate

Einzelteilzeichnung / Component Drawing

Werkstoff: 3.1645
Material:

●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:

HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany

Created	Responsible	Released
M-TS		
05.09.2017		

Version	Revision	Sheet	Page
D1226664-00	A	01	1 of 1

Document number

19.4.2 Drawing a sample contour

To draw the displayed contour:

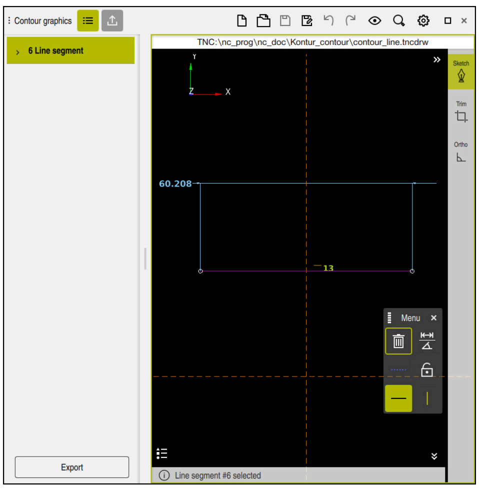
- Create a new contour

  Further information: "Creating a new contour", Page 563
- Configure **Contour settings**

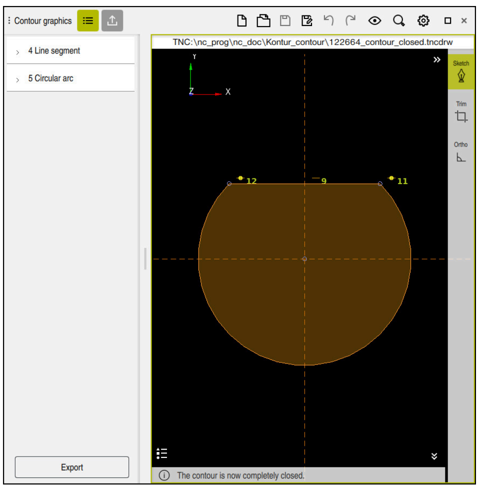
> In the **Contour settings** window, you can define basic settings for drawing. For this example, you can use the default settings.

Further information: "The Contour settings window", Page 562

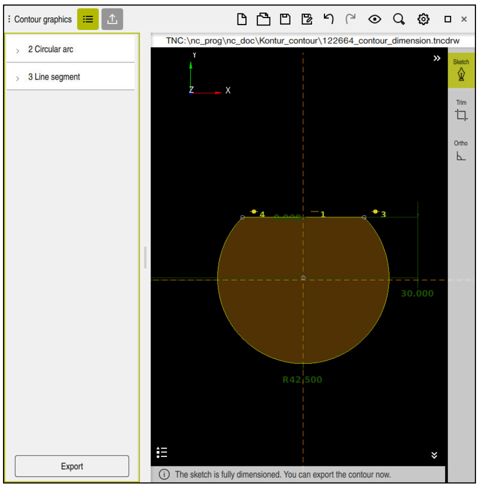
- Draw a horizontal **Line segment**
- Select the end point of the drawn line
- > The control shows the X and Y distance of the line to the center.
- Enter Y distance to center (e.g., **30**)
- > The control positions the line according to the condition set.
- Draw a **Circular arc** from one end point 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

19.4.3 Exporting a drawn contour

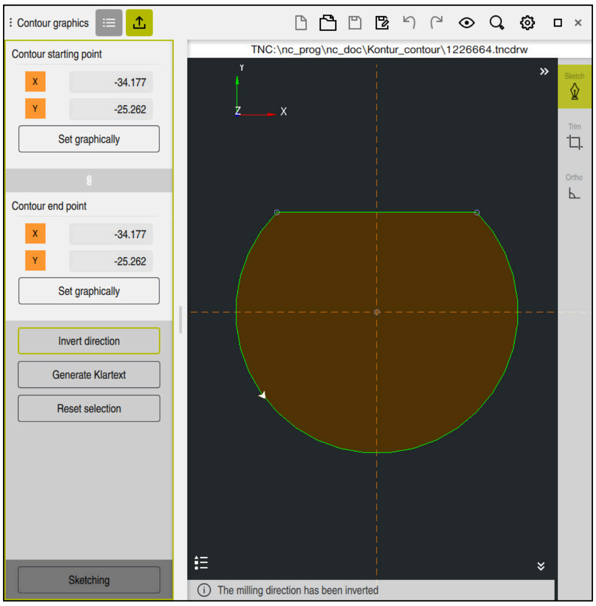
To export the drawn contour:

- 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.

> **i** 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**

20

ISO

20.1 Fundamentals

Application

The ISO 6983 standard defines a universal NC syntax.

Further information: "ISO example", Page 576

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: User's Manual for Setup and Program Run
- Programming ISO programs on the control

  Further information: "ISO syntax", Page 579
  - In addition to the standardized ISO syntax, you can program HEIDENHAIN-specific cycles as G functions.

    Further information: "Cycles", Page 598
  - Coding in Klartext syntax allows you to use some NC functions in ISO programs.

    Further information: "Klartext functions in ISO programming", Page 600
- Testing of NC programs using Simulation mode

  Further information: "The Simulation Workspace", Page 629
- Running NC programs

  Further information: User's Manual for Setup and Program Run

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 106

Contents of an NC block

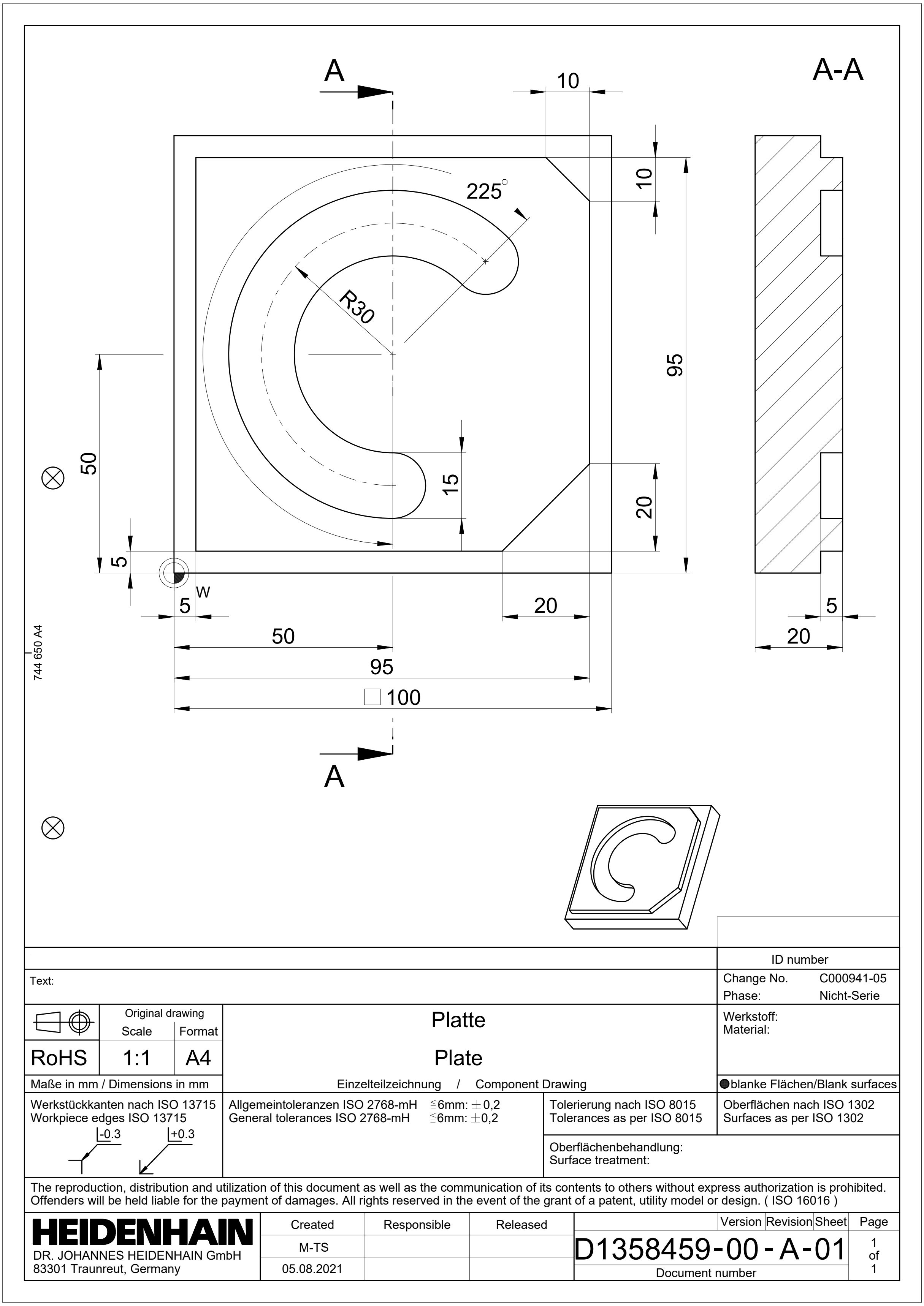
```
N110 G01 G90 X+10 Y+0 G41 F3000 M3
```

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 579
X+10 Y+0	Coordinates **Further information:** "Fundamentals of coordinate definitions", Page 154
G41	Tool radius compensation **Further information:** "Tool radius compensation", Page 590
F3000	Feed rate **Further information:** "Feed rate", Page 581
M3	Miscellaneous functions (M functions) **Further information:** "Miscellaneous Functions", Page 437

ISO example

Example task 1338459



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 122
- 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 587
  Further information: "Graphical programming", Page 555
- 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 415

20.2 ISO syntax

20.2.1 Keys

You can use the following keys to insert ISO syntax:

Key	ISO syntax	Further information
TOOL CALL	Tool call **T**	Page 580
TOOL DEF	Tool definition **G99**	Page 581
L	Straight line **G01**	Page 582
CHF	Chamfer **G24**	Page 582
RND	Rounding arc **G25**	Page 583
CC	Circular arc **G02**	Page 584
C	Circular arc **G03**	Page 584
CR	Circular arc **G05**	Page 584
CT	Tangential arc **G06**	Page 585
LBL SET	Label **G98**	Page 586
LBL CALL	Subprogram call and program-section repeat **L**	Page 587 Page 587
STOP	Stop in the NC program **G38**	Page 590

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 132

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 580

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 135
- Rotationally symmetric workpiece blank with **BLK FORM ROTATION**

  Further information: "Rotationally symmetric workpiece blank with BLK FORM ROTATION", Page 136
- STL file as workpiece blank with **BLK FORM FILE**

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 137

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 144

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 580

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 148

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 149

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**!

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 150

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 150

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.

> 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 162

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 182

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 164

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 166

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 168

> 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 179

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 181

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 170

> When you program a radius **R**, there is no need to define a circle center.
> **Further information:** "Circular path with a defined radius", Page 585

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 185

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 172

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 175

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 187

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 195

You can also use polar coordinates when programming the **APPR** and **DEP** NC functions.

Further information: "Approach and departure functions with polar coordinates", Page 209

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 222

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 222

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 223

> 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 587

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 225

Selection functions

Further information: "Selection functions", Page 226

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 226

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 257

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.

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 555

%:CNT corresponds to the **SEL CONTOUR** Klartext syntax.

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 226

Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 228

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: User's Manual for Machining Cycles

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: User's Manual for Machining Cycles

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 326

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 437

G38 corresponds to the **STOP** Klartext syntax.

Further information: "Miscellaneous functions M and the STOP function ", Page 438

Programming variables

The control provides the following options for programming variables in ISO programs:

Function group	Further information
Basic arithmetic operations	Page 592
Trigonometric functions	Page 593
Circle calculations	Page 594
Jump commands	Page 595
Special functions	Page 597
String functions	Corresponds to the Klartext syntax Page 522
Counters	Corresponds to the Klartext syntax Page 531
Calculations using formulas	Corresponds to the Klartext syntax Page 517
Function for the definition of complex contours	Corresponds to the Klartext syntax See the User's Manual for Machining Cycles

The control distinguishes between the **Q**, **QL**, **QR**, and **QS** variable types (parameter types).

Further information: "Variable Programming", Page 479

> 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 532

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 494

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 517

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 = √(5²+4²)
```

```
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 496

> 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 517

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 498

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: "The Jump commands folder", Page 500

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 513
D27	Write to a freely definable table **Further information:** "Writing to a freely definable table with FN 27: TABWRITE", Page 513
D28	Read from a freely definable table **Further information:** "Reading a freely definable table with FN 28: TABREAD", Page 515

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 501 **Further information:** "Preassigned error numbers for FN 14: ERROR", Page 709
D16	Output formatted texts **Further information:** "Outputting text formatted with FN 16: F-PRINT", Page 502
D18	Read system data **Further information:** "Read system data with FN 18: SYSREAD", Page 509 **Further information:** "System data", Page 714
D19	Transfer values to the PLC **Further information:** "Special functions defining the machine behavior", Page 708
D20	Synchronize NC and PLC **Further information:** "Special functions defining the machine behavior", Page 708
D29	Transfer values to the PLC **Further information:** "Special functions defining the machine behavior", Page 708
D37	Create user-defined cycles **Further information:** "Special functions defining the machine behavior", Page 708
D38	Send information from the NC program **Further information:** "Sending information from the NC program with FN 38: SEND", Page 510

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

20.3 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: User's Manual for Machining Cycles

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 274

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 587
- If you do not use **%:TAB:**, you have to activate the datum table manually.

  Further information: "Activating the datum table manually", Page 256

20.4 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**
- NC functions for coordinate transformations: **TRANS DATUM**, **TRANS MIRROR**, **TRANS ROTATION**, and **TRANS SCALE**

  Further information: "NC functions for coordinate transformation", Page 258
- File functions: **FUNCTION FILE** and **OPEN FILE**

  Further information: "Programmable file functions", Page 368
- Functions for machining with parallel axes: **PARAXCOMP** and **PARAXMODE**

  Further information: "Working with the parallel axes U, V and W", Page 408
- Programs that use normal vectors

  Further information: "CAM-generated NC programs", Page 422
- Table access with SQL statements

  Further information: "Table access with SQL statements", Page 532
- Changing kinematics with **WRITE KINEMATICS**

21

User aids

21.1 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 37
- User's Manual as the **TNCguide** integrated product aid
  Further information: "User's Manual as integrated product aid: TNCguide", Page 36

Description of function

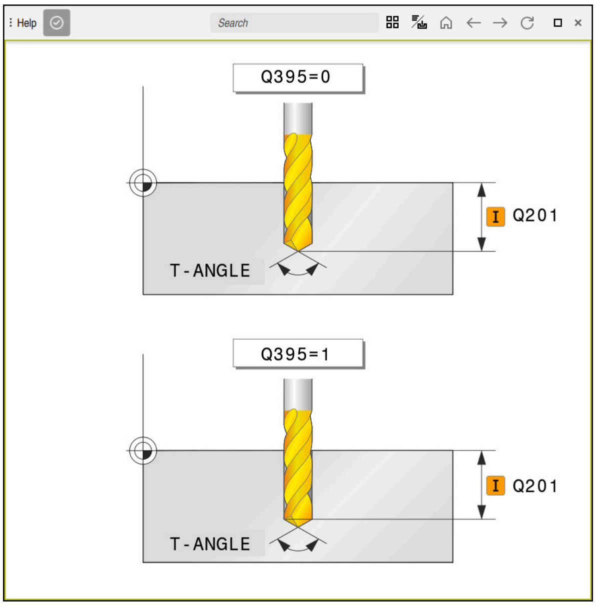
The **Help** workspace can be selected in the **Editor** operating mode and in the **MDI** application.

Further information: "The Editor operating mode", Page 109

Further information: User's Manual for Setup and Program Run

While the **Help** workspace is active, the control displays the help graphic there and not in a pop-up window.

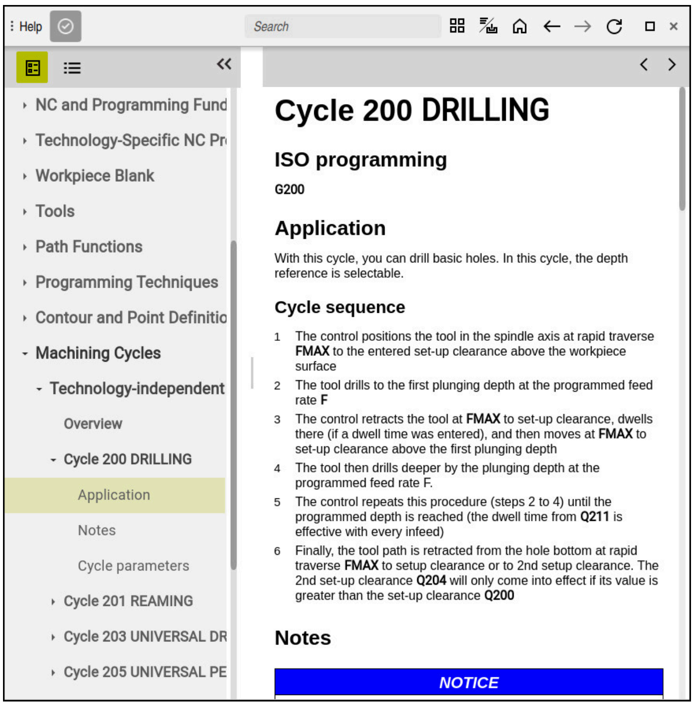
Further information: "Help graphic", Page 113



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 36



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 39
	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 38
	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 37
← →	**Navigate** Navigate between the contents opened recently
	Refresh

TNCguide has additional icons.

Further information: "User's Manual as integrated product aid: TNCguide", Page 36

21.2 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

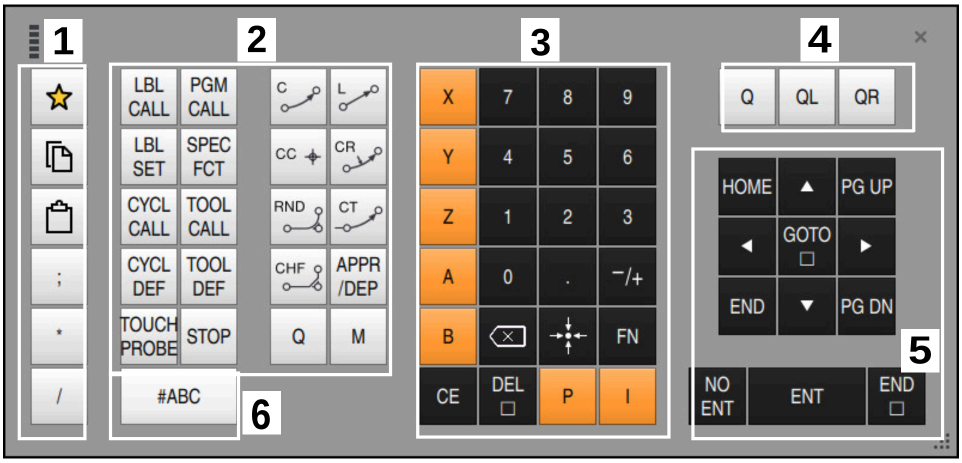
The control opens NC input mode by default after the start procedure.

You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

NC input areas



Virtual keyboard in NC input mode

NC input mode contains the following areas:

1 File functions
- Define favorites
- Copy
- Paste
- Add comment
- Add structure item
- Hide NC block

2 NC functions

3 Axis keys and numerical input

4 Q parameters

5 Navigation and dialog keys

6 Switch to text input

If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
- **Q**
- **QL**
- **QR**

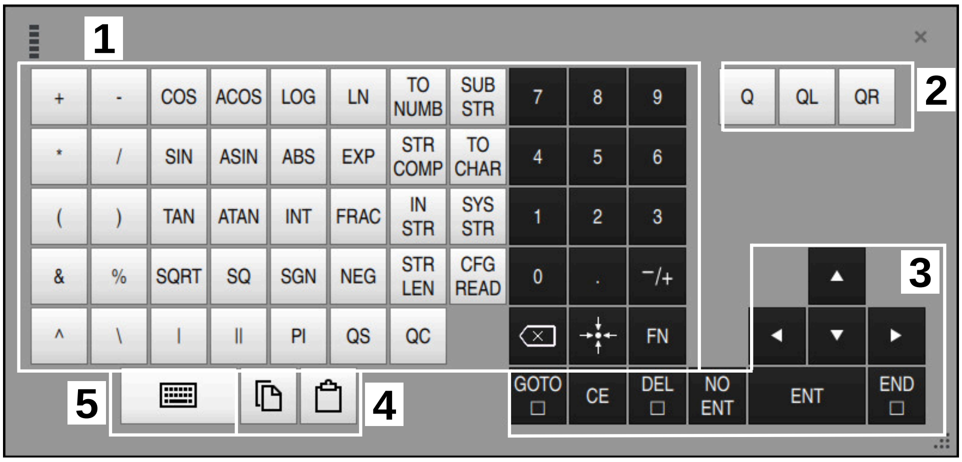
Text input areas

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

21.2.1 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.

21.3 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.

21.3.1 Selecting an NC block with GOTO

To select an NC block:

GOTO
☐

- Select **GOTO**
- > The control opens the **GOTO jump instruction** window.
- Enter the block number

OK

- 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: User's Manual for Setup and Program Run

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**.

21.4 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 118

21.4.1 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

21.4.2 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

21.4.3 Commenting an NC block out or in

Use the **Comment out/in** button to define an existing NC block as a comment or to change a comment back to an NC block.

To comment an existing NC block in or out:

- Select the desired NC block

- Select **Comment Off/On**
- > The control inserts a **;** character at the beginning of the block.
- > If the NC block is already defined as a comment, the control removes the **;** character.

21.5 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: User's Manual for Setup and Program Run

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.

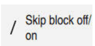
If the toggle switch is active, then the control dims the NC blocks to be skipped.

Further information: User's Manual for Setup and Program Run

21.5.1 Hiding or showing NC blocks

To hide or show an NC block:

- Select the desired NC block



- 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.

21.6 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 610

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 610

21.6.1 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

21.7 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.

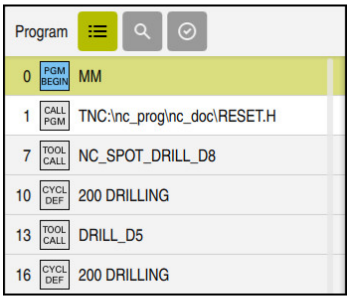
Related topics

- The **Program** workspace, defining contents of the **Structure** column

  Further information: "Settings in the Program workspace", Page 113
- Inserting structure items manually

  Further information: "Structuring of NC programs", Page 610

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 113

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.

> 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 608

21.7.1 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.

21.7.2 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 618
- 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.

> 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.

21.8 The Search column in the Program workspace

Application

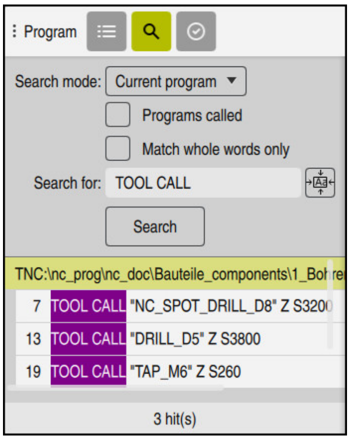
In the **Search** column, you can search the NC program for any character strings, such as individual syntax elements. The control lists all the results found.

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 120

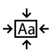
Description of function



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 615
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.
	Use the **Apply selection** icon to transfer the currently selected syntax element to the input area. If the selected NC block is not edited, the control accepts the syntax 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.

21.8.1 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**)

- ▶ 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.

- ▶ 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.

21.9 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.

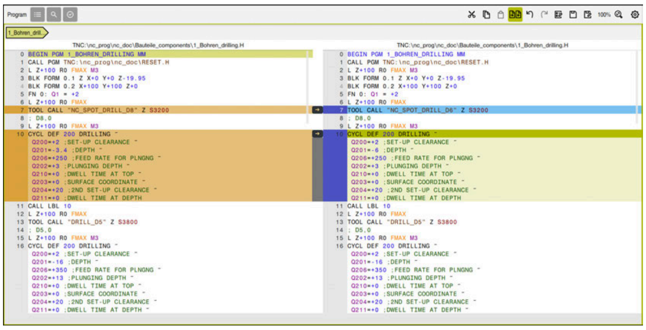
Requirements

- 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 106

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.

21.9.1 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 618

21.10 Context menu

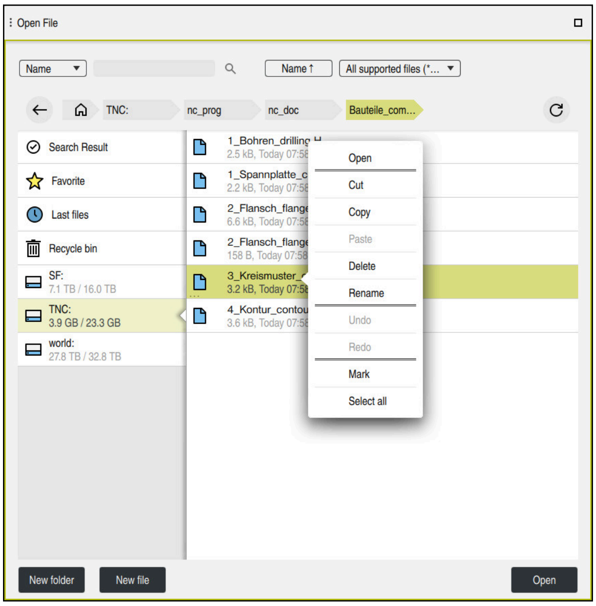
Application

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**

> 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 74

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

> 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 618

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**

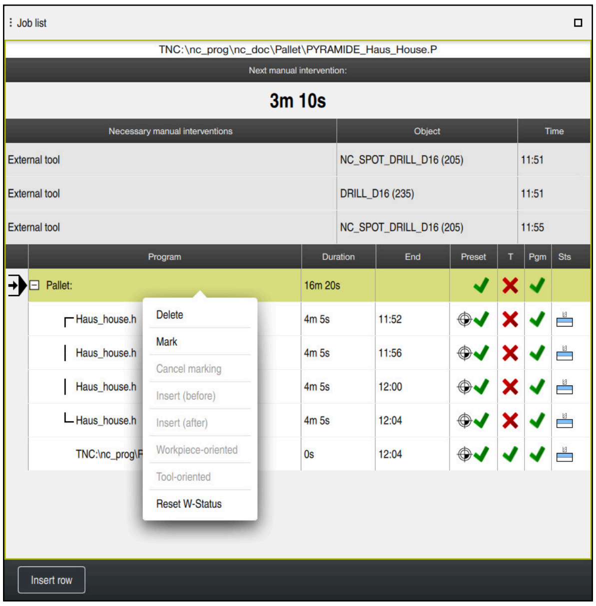
  The control inserts the row instead of the currently selected table row.
- **Append**

  The control appends the row at the end of the table.

  > 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 672

Context menu in the Job list workspace



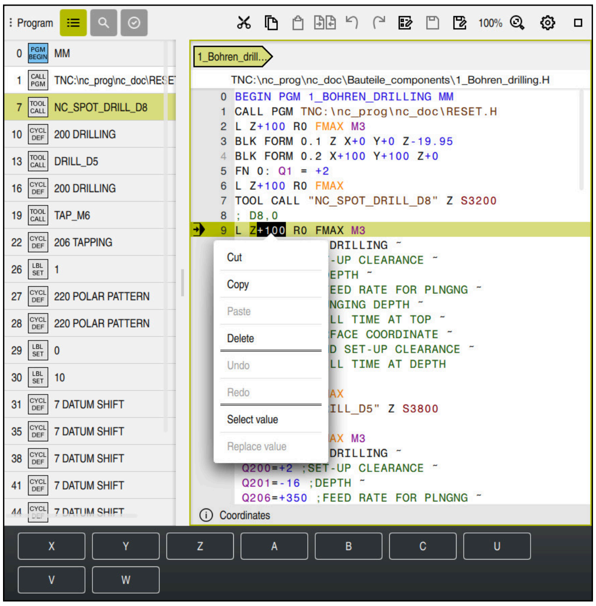
Context menu in the **Job list** workspace

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 654

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 230
- **Edit contour**

  Only in the **Editor** operating mode

  Further information: "Importing contours into graphical programming", Page 564
- **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 110

> 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 623

Further information: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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 122

21.11 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.

21.11.1 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.

21.11.2 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.

21.11.3 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.

21.12 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.

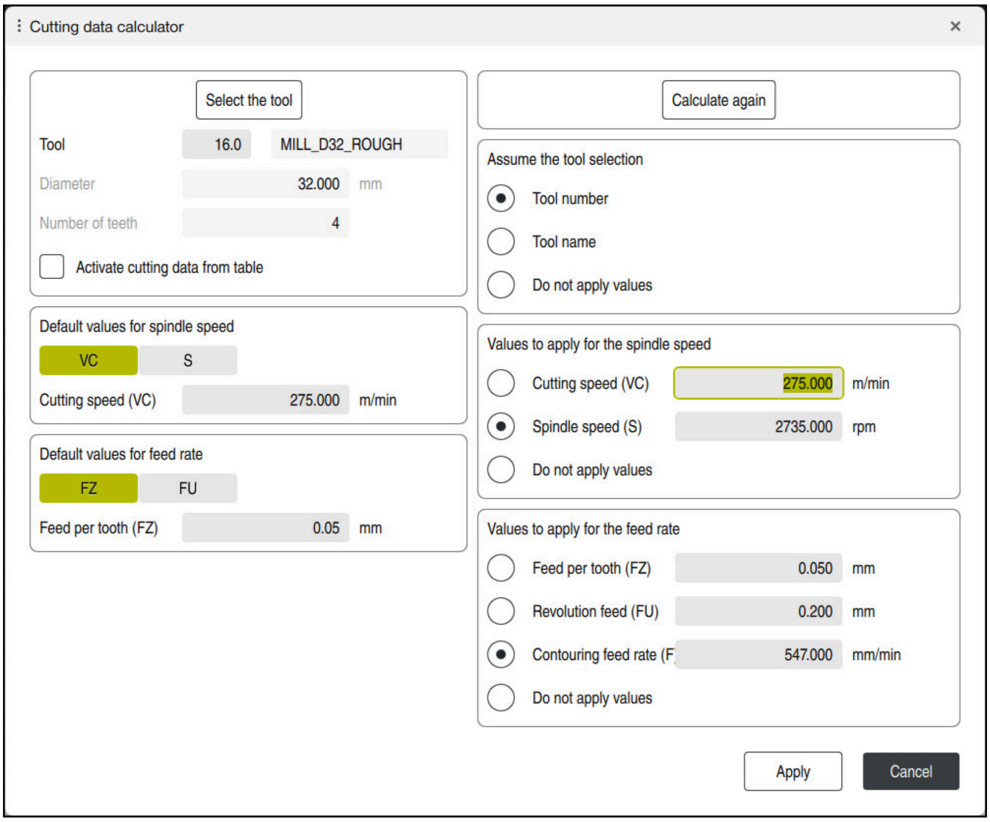
In OCM cycles (#167 / #1-02-1), the **OCM cutting data calculator** is available.

Further information: User's Manual for Machining Cycles

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 626

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 697
- Tool cutting material in table **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 697
- 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 698
  >
  > 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 699

- 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

  Further information: User's Manual for Setup and Program Run

21.12.1 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.

21.12.2 Calculating the cutting data with tables

The following prerequisites must be fulfilled in order to calculate the cutting data with tables:

- The **WMAT.tab** table exists
- The **TMAT.tab** table exists
- The ***.cut** or ***.cutd** table exists
- Tool material and cutting data table are assigned in the tool management

To calculate the cutting data with tables:

- Edit the desired NC block
- Open the **Cutting data calculator**
- Select **Activate cutting data from table**
- Use **Select material** to choose the workpiece material
- Use **Select type of machining** to choose the combination of workpiece material and tool material
- Select the desired values to be applied

Apply

- Select **Apply**
- > The control applies the calculated values in the NC block.

22

The Simulation Workspace

22.1 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 132

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.

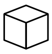
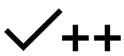
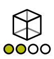
Further information: "The Simulation settings window", Page 636

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**

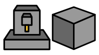
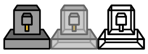
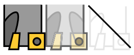
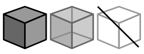
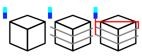
Icons in the Simulation workspace

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 632
	Open or close the **Workpiece options** column **Further information:** "The Workpiece options column", Page 634
	Open or close the **Pre-defined views** selection menu **Further information:** "Pre-defined views", Page 640
	**Save as** Export simulated workpiece as STL file **Further information:** "Exporting a simulated workpiece as STL file", Page 641
	Open or close the **Simulation settings** window **Further information:** "The Simulation settings window", Page 636
 	Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) DCM active DCM inactive **Further information:** "The Visualization options column", Page 632
	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 386
	Status of the **Advanced checks** function **Further information:** "The Visualization options column", Page 632
	Model quality **Further information:** "The Simulation settings window", Page 636
T 0	Number or name of the current tool 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. **Further information:** User's Manual for Setup and Program Run	■ The **Editor** operating mode
	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. You can choose between the following display modes for the tool paths: ■ **None**: Do not show tool paths ■ **Feed**: Show tool paths with programmed feed rate ■ **Feedrate + FMAX**: Show tool paths with programmed feed rate and with programmed rapid traverse	■ The **Workpiece** mode ■ The **Editor** operating mode
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. **Further information:** "Dynamic Collision Monitoring (DCM) in the Editor operating mode", Page 378	■ The **Editor** operating mode ■ Simulation reset or not started yet

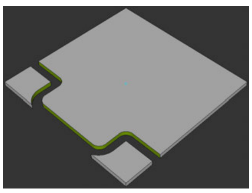
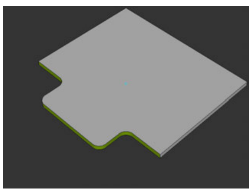
Icon or toggle switch	Meaning	Requirements
Advanced checks	If the **Advanced checks** toggle switch is active, the following checks can be performed: - **Rapid-traverse cut** - **Workpiece collision** - **Fixture collision** **Further information:** "Advanced checks in the simulation", Page 388	- The **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: User's Manual for Setup and Program Run
	- **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 609
	If the toggle switch is active, then the control dims the NC blocks to be skipped.
	Further information: "Appearance of the NC program", Page 112
	- **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 439
	If the toggle switch is inactive, then the control dims the **M1** syntax element.
	Further information: "Appearance of the NC program", Page 112
		- 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 643	■ 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 645	■ 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 645	
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 636	■ The **Editor** operating mode

Toggle switch or button	Meaning	Requirements
Workpiece coloring	■ **Grayscale** The control displays the workpiece in various shades of gray. ■ **Tool based** The control displays the workpiece in color. Each cutting tool is assigned a separate color. ■ **Model comparison** The control displays a comparison between the workpiece blank and the finished part. **Further information:** "Model comparison", Page 647 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 404 **Further information:** User's Manual for Machining Cycles	■ Type of model: **2.5D** ■ **Model comparison** function in the **Workpiece** mode only ■ **Monitoring** function in 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 machin-ing.  Workpiece before clean-up  Workpiece after clean-up	■ The **Editor** operating mode ■ Type of model: **3D**

The Simulation settings window

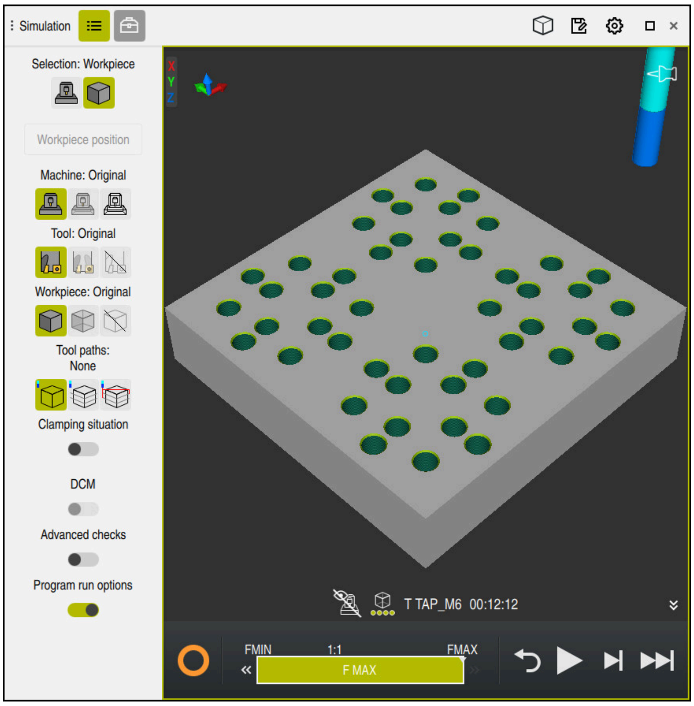
The **Simulation settings** window is available only in the **Editor** operating mode.

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 137 ■ **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:** User's Manual for Setup and Program Run

Area	Function
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 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 634 ■ **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:** User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

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 649
	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 650

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 324

  Further information: User's Manual for Setup and Program Run

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

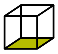
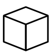
22.2 Pre-defined views

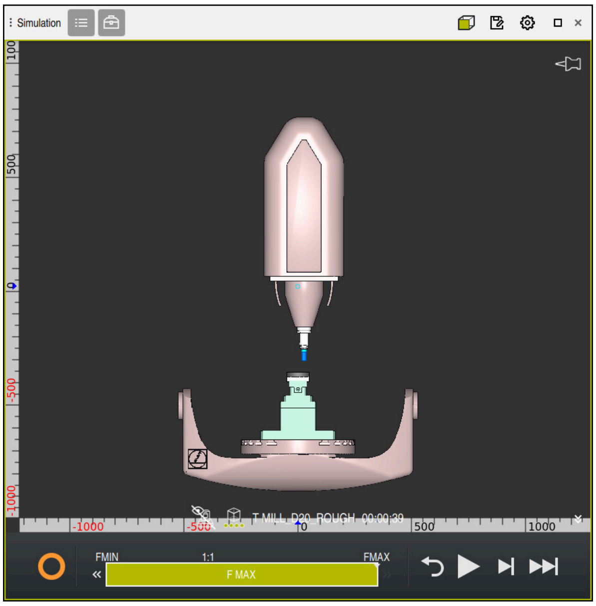
Application

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

22.3 Exporting a simulated workpiece as STL file

Application

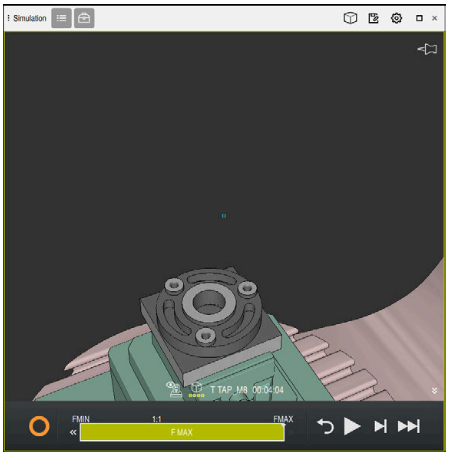
In the simulation you can use the **Save** function to save the current status of the simulated workpiece as a 3D model in STL format.

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 137
- Customizing STL files in **CAD Viewer** (#152 / #1-04-1)
  Further information: User's Manual for Setup and Program Run

Description of function



Simulated workpiece

This function can be used only in the **Editor** operating mode.

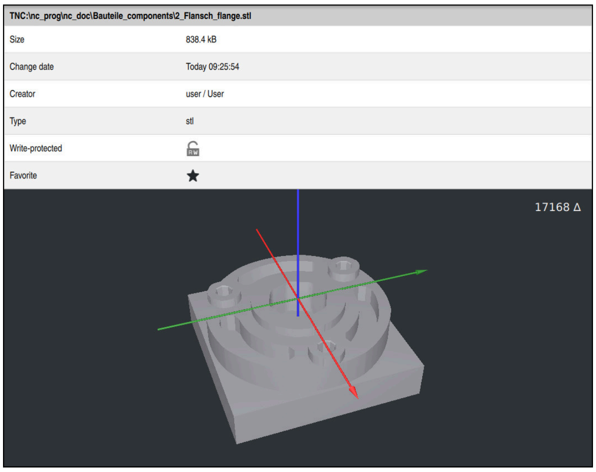
The control can only display STL files with up to 20,000 triangles. If the exported 3D model has too many triangles, due to the excessively high model quality, then you cannot use the exported 3D model on the control.

In this case, reduce the model quality in the simulation.

Further information: "The Simulation settings window", Page 636

You can also use the **3D mesh** function to reduce the number of triangles (#152 / #1-04-1).

Further information: User's Manual for Setup and Program Run



Simulated workpiece as saved STL file

22.3.1 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 636

22.4 Measuring function

Application

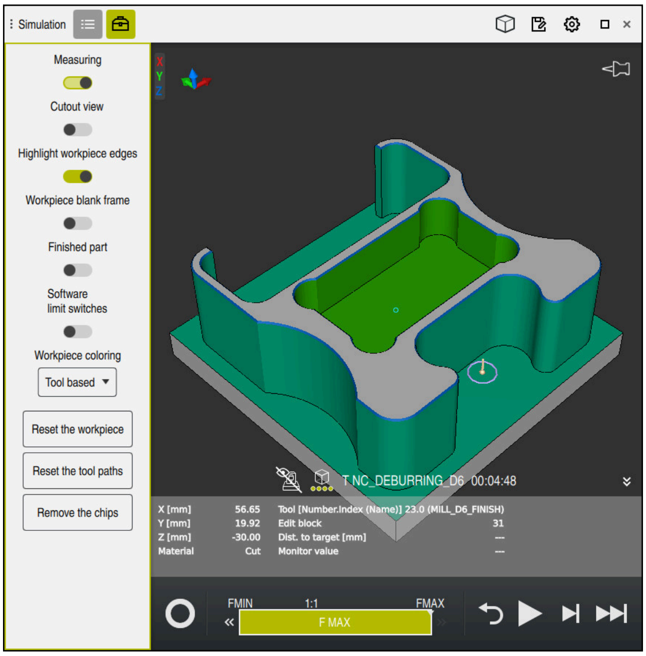
Use the measuring function to measure any points on the simulated workpiece. The control shows various pieces of information about the measured surface.

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 243
- 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: User's Manual for Setup and Program Run

22.4.1 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
- 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.

22.5 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

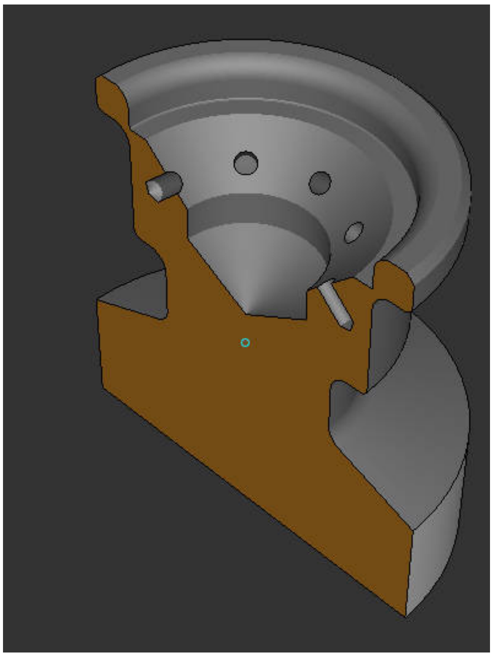
The Cutout view can be used in the **Editor** mode only.

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.

22.5.1 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**

22.6 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

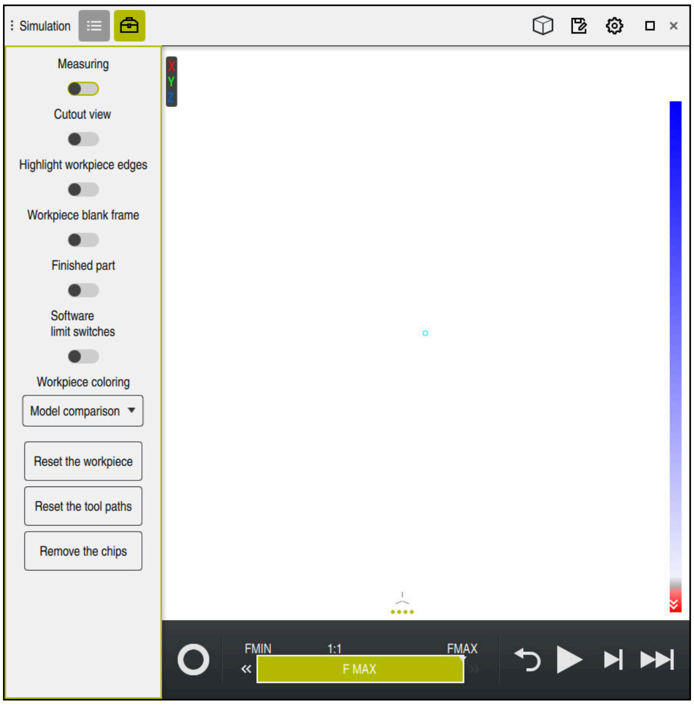
- Programming the blank and finished part with STL files

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 137

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 645

22.7 Center of rotation in the simulation

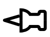
Application

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.

22.7.1 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".

22.8 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**)

22.9 Simulating an NC program up to a certain NC block

Application

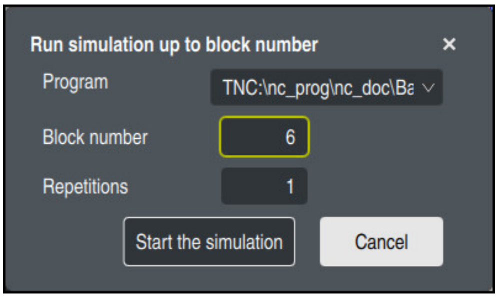
If you want to check a critical point in the NC program then you can simulate the NC program up to a specific NC block that you specify. Once the NC block is reached in the simulation, the control stops the simulation automatically. Starting from this NC block you can then continue the simulation, for example in **Single Block** mode or at a lower simulation speed.

Related topics

- Possibilities in the action bar
  Further information: "Action bar", Page 638
- Simulation speed
  Further information: "Simulation speed", Page 649

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

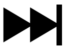
The following settings options are offered in the **Run simulation up to block number** window:

- **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 225

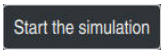
22.9.1 Simulating an NC program up to a certain NC block

To simulate up to a specific NC block:

- Open the **Simulation** workspace



- Select **Run simulation up to block number**
- > The control opens the **Run simulation up to block number** window.
- Use the selection menu in the **Program** field to specify the main program or called program
- Enter the number of the desired NC block in the **Block number** field
- If the block involves a program-section repeat, enter the number of the iteration of the program-section repeat in the **Repetitions** field



- Select **Start the simulation**
- > The control simulates the workpiece up to the selected NC block.

23

Pallet Machining and Job Lists

23.1 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 664

23.1.1 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 700

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 655

23.2 The Job list workspace

23.2.1 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 700
- The **Form** workspace for pallets
  Further information: "The Form workspace for pallets", Page 662
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 664

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 655

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 657

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 656
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 654 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 656
Sts	Machining status **Further information:** "Machining status", Page 656

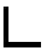
Status of the pallet, fixture, or NC program

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

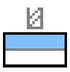
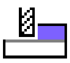
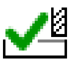
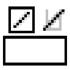
The control uses the following icons to display the machining method:

Icon	Meaning
No symbol	Workpiece-oriented machining
 	Tool-oriented machining ■ Start ■ End

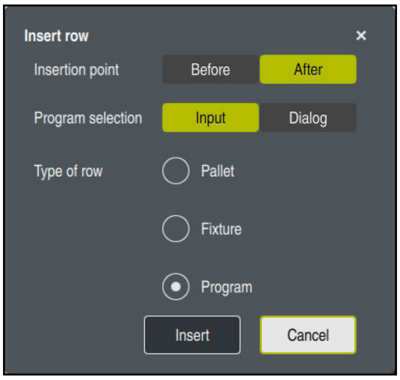
Machining status

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 662

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 658

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.

23.2.2 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 654
- Editing a pallet table in the **Form** workspace

  Further information: "The Form workspace for pallets", Page 662
- Contents of the pallet table

  Further information: "Pallet table *.p", Page 700

Requirements

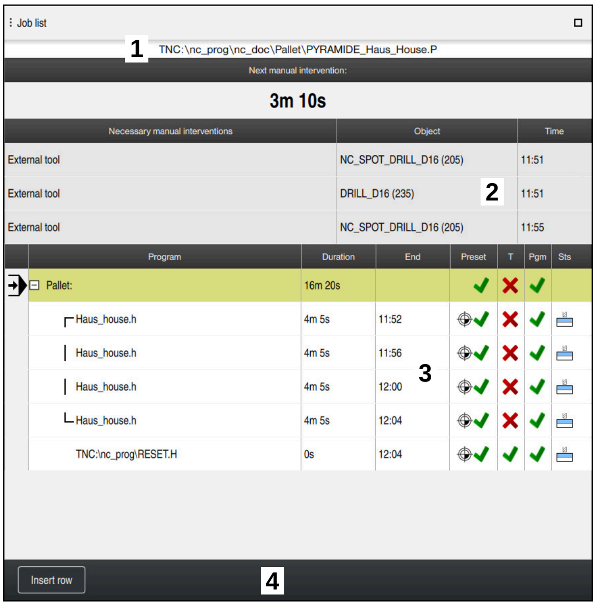
- Batch Process Manager software option (#154 / #2-05-1)

  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: User's Manual for Setup and Program Run

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 661

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.

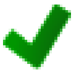
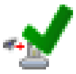
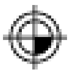
Information about the pallet table

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 656
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 654 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 656
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 662
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 662
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 662
Sts	Machining status **Further information:** "Machining status", Page 656

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 .

23.3 The Form workspace for pallets

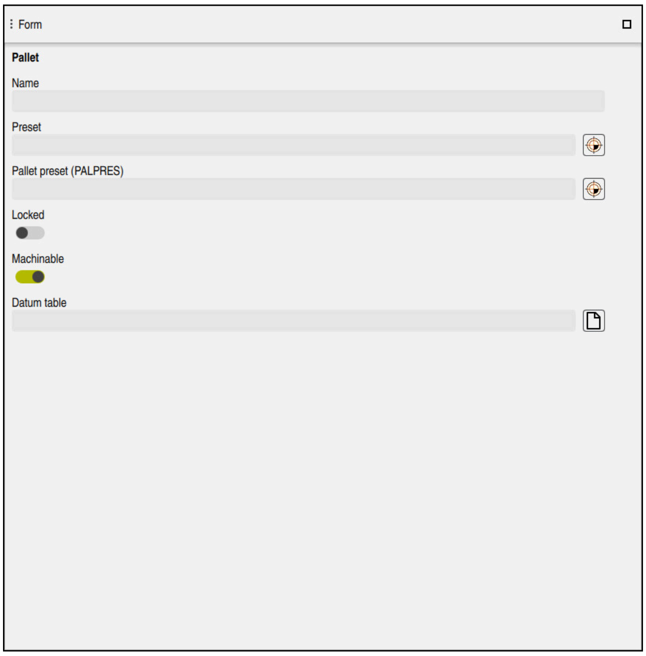
Application

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 654
- Contents of the pallet table
  Further information: "Pallet table *.p", Page 700
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 664

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.

23.4 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 700
- Block scan for mid-program startup in a pallet table
  Further information: User's Manual for Setup and Program Run

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 315
  - **M144** (#9 / #4-01-1)
    Further information: "Taking the tool offset into account in calculations M144 (#9 / #4-01-1)", Page 469
  - **M101**
    Further information: "Automatically inserting a replacement tool with M101", Page 473
  - **M118** (#21 / #4-02-1)
    Further information: "Activating handwheel superimpositioning with M118 (#21 / #4-02-1)", Page 453
  - Changing the pallet preset
    Further information: "Pallet preset table", Page 669

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 700

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 618

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 700

> 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.

23.5 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 700
- Workpiece preset management

  Further information: User's Manual for Setup and Program Run

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.

You can check the active pallet preset and the defined values in the **Setup** application.

Further information: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

24

Tables

24.1 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 361

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "The Open File workspace", Page 360

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 676

Further information: "The Form workspace for tables", Page 682

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 618

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:** ■ **Line number (Tool number?)** ■ **Number of rows** ■ **Index** ■ **Append** Append rows at the end of the table **Further information:** User's Manual for Setup and Program Run
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:** User's Manual for Setup and Program Run
Import	The control imports tool data.
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:** User's Manual for Setup and Program Run
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes.

Refer to your machine manual.
If necessary, the machine manufacturer adapts the buttons.

24.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Enable **Editing**
- > The control enables the values for editing.

> 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.

> 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 350

  > 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.

24.2 The Create new table window

Application

You can create tables using the **Create new table** window in the **Quick selection new table** workspace.

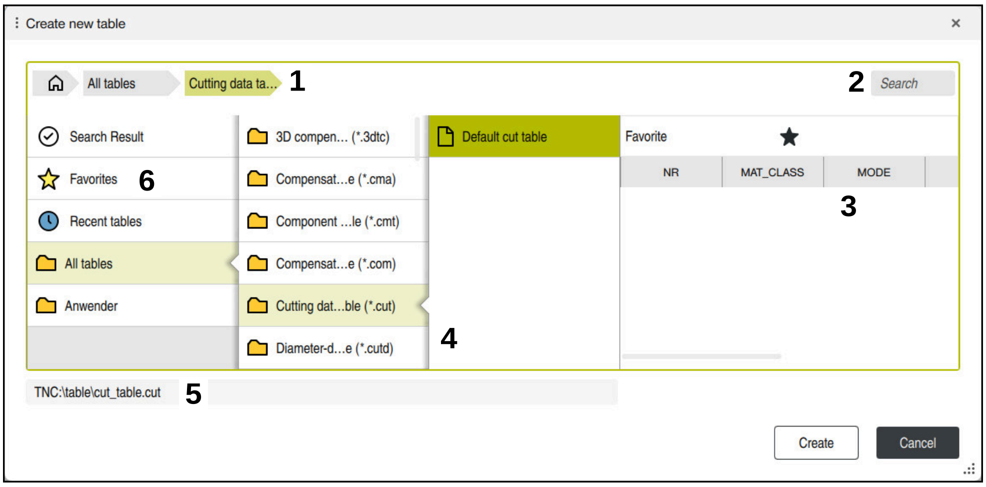
Related topics

- The **Quick selection new table** workspace

  Further information: "Quick selection workspaces", Page 361
- Available file types for tables

  Further information: "File types", Page 356

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

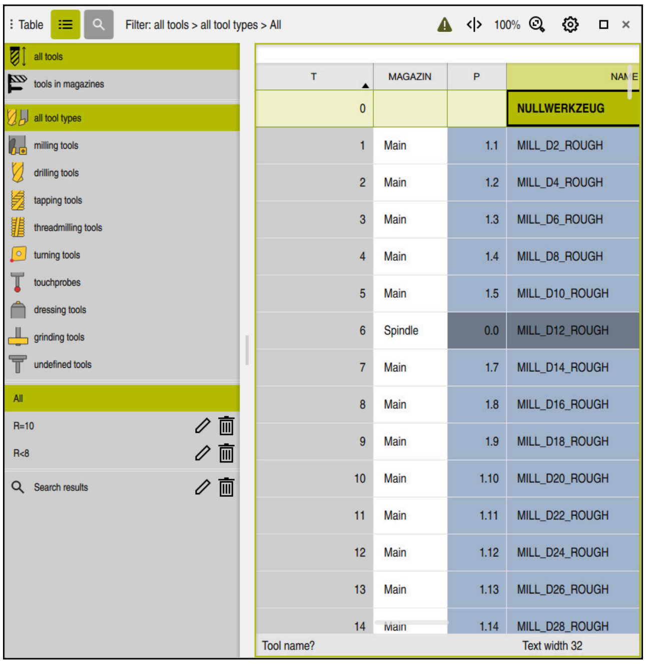
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer 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**).

24.3 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 677
CTRL + F	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 680
	Enable or disable **Change column width**
	Edit table characteristics **Further information:** "Modifying the properties of freely definable tables", Page 692
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 680
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 680
	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

> 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:

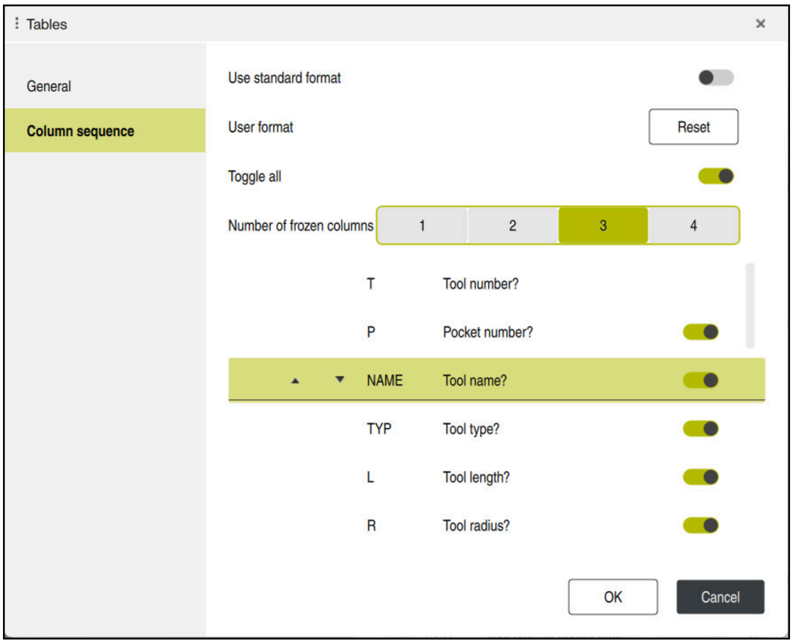
- **General**
- **Column sequence**

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.

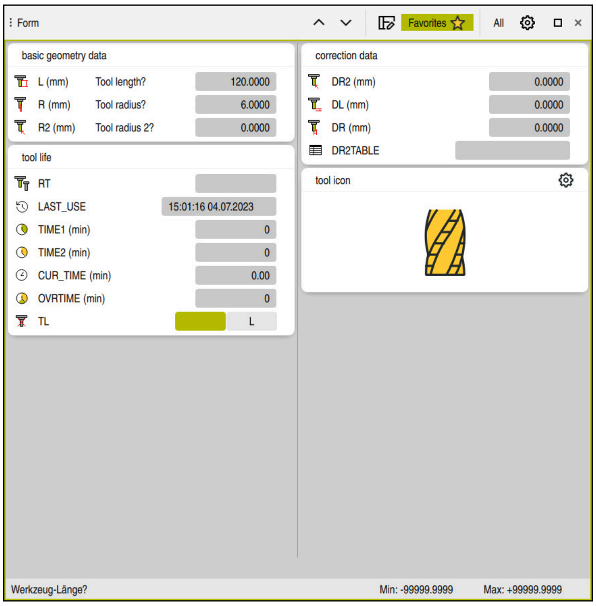
The settings in the **Column sequence** area only apply to the currently opened table.

24.4 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).

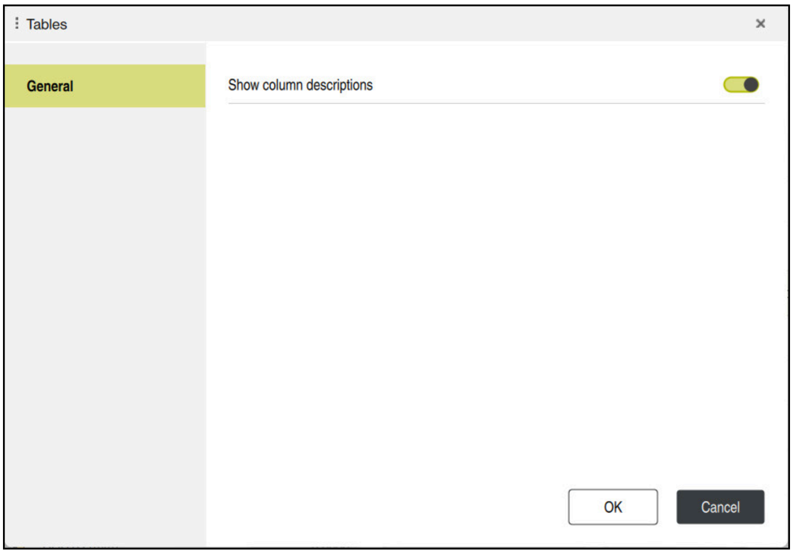
Buttons and icons

The **Form** workspace contains the following buttons, icons or shortcuts:

Buttons, icons or shortcuts	Meaning
^ SHIFT + UP v 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 684 ■ 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 684 ■ 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.



24.4.1 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: User's Manual for Setup and Program Run

24.5 Accessing table values

24.5.1 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 686
TABDATA WRITE	Write a value to a table cell	Page 687
TABDATA ADD	Add a value to a table value	Page 689

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 480
- Tool table

  Further information: User's Manual for Setup and Program Run
- Compensation tables

  Further information: "Compensation tables", Page 704
- Reading values from freely definable tables

  Further information: "Reading a freely definable table with FN 28: TABREAD", Page 515
- Writing values to freely definably tables

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 513

24.5.2 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

24.5.3 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

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

24.5.4 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 331

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

24.6 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 513

Description of function

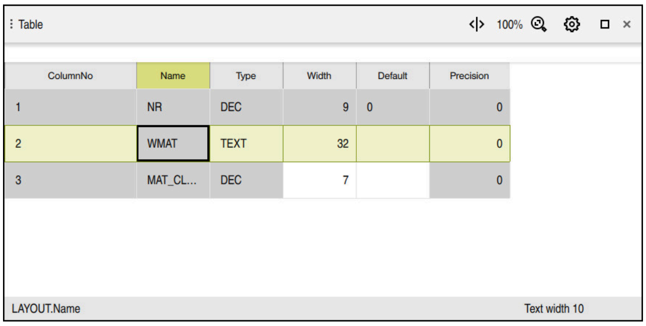
When you create a freely definable table, the control will provide various table templates for selection.

The machine manufacturers can create their own table templates and store them in the control.

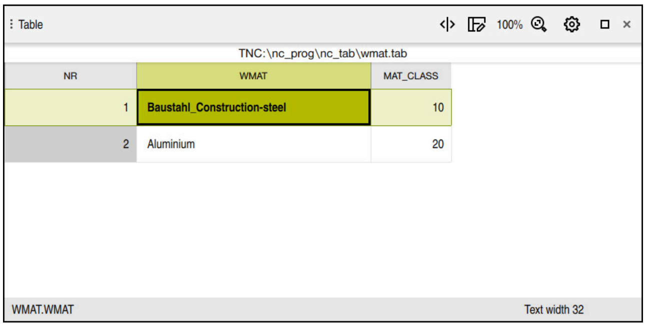
After you have created a freely definable table, you can modify its properties. you modify the table properties in the **LAYOUT** application.

Further information: "Modifying the properties of freely definable tables", Page 692

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:

<table>
<tr><th>Column</th><th>Meaning</th></tr>
<tr><td>Name</td><td>Name of the column</td></tr>
<tr><td>Width</td><td>Maximum number of characters in the column</td></tr>
<tr><td>Default</td><td>Default value of each new row
Optional input</td></tr>
<tr><td>Type</td><td>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.</td></tr>
<tr><td>Precision</td><td>Maximum number of decimal places</td></tr>
</table>

24.6.1 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.

> You cannot change the column name or column type later.

- Select the desired column type

  Further information: "Properties of a table column", Page 691
- 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 532
- 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 680

24.7 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: User's Manual for Machining Cycles

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**

24.7.1 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

- ▶ 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: User's Manual for Setup and Program Run

24.8 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 694
- Editing a datum table during a program run

  Further information: User's Manual for Setup and Program Run
- Preset table

  Further information: User's Manual for Setup and Program Run

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: User's Manual for Setup and Program Run

Parameters in datum tables

A datum table provides the following parameters:

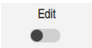
Parameter	Meaning
D	Row number in the datum table Input: **0...99999999**
X	X coordinate of the datum Transformation relating to the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 243 Input: **-99999.99999...+99999.99999**
Y	Y coordinate of the datum Transformation relating to the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 243 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 243 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 238 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 238 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 238 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 238 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 238 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 238 Input: **-99999.99999...+99999.99999**
DOC	**Comment on shift?** Input: **Text width 15**

24.8.1 Editing a datum table

You can edit the active datum table during program run.

Further information: User's Manual for Setup and Program Run

To edit a datum table:



- 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

24.9 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 697
- Table for tool materials **TMAT.tab**
  Further information: "Table for tool materials TMAT.tab", Page 697
- Cutting data table ***.cut**
  Further information: "Cutting data table *.cut", Page 698
- Diameter-dependent cutting data table ***.cutd**
  Further information: "Diameter-dependent cutting data table *.cutd", Page 699

Related topics

- Cutting data calculator
  Further information: "Cutting data calculator", Page 625
- Tool management
  Further information: User's Manual for Setup and Program Run

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 697 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 697 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 148 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 149 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 697 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 697 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 148 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 149 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.

24.10 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 654
- Tool-oriented machining

  Further information: "Tool-oriented machining", Page 664

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:** User's Manual for Setup and Program Run 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**

Parameter	Meaning
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?** 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 664 Input: No value, **BLANK**, **INCOMPLETE**, **ENDED**, **EMPTY**, **SKIP**
PALPRES	**Pallet preset** Row number in the pallet preset table for the pallet preset to be activated Only required if a pallet preset table has been created on the control. Selection by means of a selection window Input: **-1...+999**
DOC	Comment Input: **Text width 15**

Parameter	Meaning
METHOD	**Machining method?** Machining method The control differentiates between the following entries ■ WPO: Workpiece oriented (standard) ■ TO: Tool oriented (first workpiece) ■ CTO: Tool oriented (further workpieces) **Further information:** "Tool-oriented machining", Page 664 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 664 Input: **Text width 8**
SP-X	**Clearance height?** Clearance height in the X axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 664 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 664 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 664 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 664 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 664 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 664 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 664 Input: **-999999.99999...+999999.99999**

Parameter	Meaning
SP-V	**Clearance height?** Clearance height in the V axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 664 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 664 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 654 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 654 Input: **0...99999**

24.11 Compensation tables

24.11.1 Overview

The control provides the following compensation tables:

Table	Further information
Compensation table ***.tco** Compensation in the tool coordinate system **T-CS**	Page 704
Compensation table ***.wco** Compensation in the working plane coordinate system **WPL-CS**	Page 706

24.11.2 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 329
- Contents of the compensation table ***.wco**

  Further information: "Compensation table *.wco", Page 706
- Editing compensation tables during program run

  Further information: User's Manual for Setup and Program Run
- Tool coordinate system **T-CS**

  Further information: "Tool coordinate system T-CS", Page 249

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**

24.11.3 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 329
- Contents of the compensation table ***.tco**
  Further information: "Compensation table *.tco", Page 704
- Editing compensation tables during program run
  Further information: User's Manual for Setup and Program Run
- Working plane coordinate system **WPL-CS**
  Further information: "Working plane coordinate system WPL-CS", Page 244

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**

25

Overviews

25.1 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

25.2 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 501

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

25.3 System data

25.3.1 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**.)

> The control always outputs system data in the metric system with **FN 18: SYSREAD**, regardless of the unit of the NC program.

Further information: "Read system data with FN 18: SYSREAD", Page 509

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)**).

Further information: "Read system data with SYSSTR", Page 524

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
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function resolves relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = No, 1 = Yes Only absolute directory paths are possible.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
System jump addresses				
	13	1	-	Label number or label name (string or QS) jumped to during M2/M30 instead of ending the current NC program. Value = 0: M2/M30 have the normal effect
		2	-	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
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		48	-	Tolerance
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Toler-ance)
		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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
		2	-	Radius compensation: 0 = R0 1 = RR/RL 10 = Face milling 11 = Peripheral milling
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		39	Tool no.	ACC
		40	Tool no.	Pitch for thread cycles
		41	Tool no.	AFC: reference load
		42	Tool no.	AFC: overload early warning
		43	Tool no.	AFC: overload NC stop
		44	Tool no.	Exceeding the tool life
		45	Tool no.	Front-face width of indexable insert (RCUTS)
		46	Tool no.	Usable length of the milling cutter
		47	Tool no.	Neck radius of the milling cutter (RN)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
File information				
	56	1	-	Number of lines of the tool table
		2	-	Number of lines of the active datum table
		4	-	Number of 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)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values 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 [kgm^2] (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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode. To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 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.
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 compen-

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				sation 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.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX+2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis participates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		5	Secondary axis	Read whether the given secondary axis is used in the kinematics model. −1 = Axis not in the kinematics model 0 = Axis is not included in the kinematics calculation:

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		6	Axis	Angle head: Displacement vector in the basic coordinate system B-CS through angle head Index: 1, 2, 3 (X, Y, Z)
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 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)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
		126	-	M126: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (look-ahead calculation).
		3	-	Read the processing time of the current NC program.
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		16	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm
		20	0	The current calendar week number according to ISO 8601 (real time)
			1	The current calendar week number according to ISO 8601 (look-ahead calculation)
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No Global Program Settings active 1 = Any GPS settings active
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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 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)
		9	Axis	GPS: Value for handwheel superimpositioning 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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
			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
			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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 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
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				

Group name	Group number ID...	System data number NO....	Index IDX...	Description
	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)
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	10	Read value from active point table.
			11	Read value from active point table.
			1-3 X/Y/Z	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, 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,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s^3

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		45	-	Shape of the smoothing filter (**CfgSmoothingFilter/shape**) 0 = Off 1 = Average 2 = Triangle
		46	-	Order of smoothing filter (only odd-numbered values) (**CfgSmoothingFilter/order**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		47	-	Type of acceleration profile (**CfgLaPath/profileType**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		48	-	Type of acceleration profile for rapid traverse (**CfgLaPath/profileTypeHi**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		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_IpcJerkFact**)
		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**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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
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, ...)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read Functional Safety (FS) information				
	820	1	-	FS limitations: 0 = No Functional Safety (FS) 1 = Guard door open (SOM1) 2 = Guard door open (SOM2) 3 = Guard door open (SOM3) 4 = Guard door open (SOM4) 5 = All guard doors closed
Counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		41	-	AFC: reference load
		42	-	AFC: overload early warning
		43	-	AFC: overload NC stop
		44	-	Exceeding the tool life
		45	-	Front-face width of indexable insert (RCUTS)
		46	-	Usable length of the milling cutter
		47	-	Neck radius of the milling cutter (RN)
		48	-	Radius at the tool tip (R_TIP)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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. –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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = NC program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being updated –1 = Iniprog cycle was canceled before block scan –2 = Cancellation during block scan –3 = Cancellation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancellation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last **FN 14** error
		16	-	Real execution active? 1 = execution, 0 = simulation
		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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	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

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display the **Spindle 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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
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 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.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		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
		22	-	Symbolic name of the compensation table in the WPL-CS working plane coordinate system

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in the tool call (system string)				
	10060	1	-	Tool name
Read machine kinematics (system strings)				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Traverse range switchover (system string)				
	10300	1	-	Key name of the last active range of traverse
Read current system time (system string)				
	10321	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**.
Read the data for pallet machining (system string)				
	10510	1	-	Pallet name
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				

Group name	Group number ID...	System data number NO....	Index IDX...	Description
	10630	10	-	The string corresponds to the format of the version ID shown (e.g., **340590 09** or **817601 05 SP1**)
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

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com
The Information Site for
HEIDENHAIN Controls

Klartext App
Klartext on your
mobile device

Google Play Store

Apple App Store

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:

www.heidenhain.com/products/touch-probes-and-vision-systems

Original document
1409856-20 · Ver00 · 10/2023 · H · Printed in Germany