HEIDENHAIN

TNC7

User's Manual
Programming and Testing

NC Software
81762x-19

English (en)
09/2024

Table of contents

1

New and Modified
Functions

Available documentation

> **TNC7 complete edition**
>
> The split editions of the User's Manual contain only the new and modified functions that are relevant to the corresponding User's Manual. The **complete edition** contains all new and modified functions of this software version that are relevant to the user.
>
> ID: 1369999-xx
>
> You can download this documentation free of charge from the HEIDENHAIN website.
>
> **TNCguide**

> **Overview of new and modified software functions**
>
> The additional documentation **Overview of New and Modified Software Functions** contains all new and modified functions of the current and previous software versions that are relevant to the user.
>
> ID: 1373081-xx
>
> You can download this documentation free of charge from the HEIDENHAIN website.
>
> **TNCguide**

1.1 New functions

1.1.1 Interpolating spindle (#96 / #7-04-1)

Topic	Description
Adv. Spindle Interpol. software option (#96 / #7-04-1)	The **FUNCTION SHAPING** contour planing function has been added. **Further information:** "Workpiece shaping with FUNCTION SHAPING (#96 / #7-04-1)", Page 527
	FUNCTION TURNDATA CORR is also available with the Adv. Spindle Interpol. software option (#96 / #7-04-1).
	The turning-tool table is also available with the Adv. Spindle Interpol. software option (#96 / #7-04-1).

1.1.2 About the product

Topic	Description
Remember position of windows	Use the **Remember position** icon to select whether the control will remember the position of a window the next time a window is opened.
Calculating numerical values during input	Use the **+**, **-**, *****, **/**, **(** and **)** keys for calculations within numerical input fields and table cells.
Warning if not enough main memory is available	The control must have at least 16 GB of RAM, as the control will otherwise display a warning.

1.1.3 Programming fundamentals

Topic	Description
Insert NC function window	For some NC functions, the **Insert NC function** window offers the possibility of inserting the start and end of the NC function into the NC program at the same time (e.g., **IF and END IF**).
	If you mark several NC blocks in the NC program and insert the combined NC functions, the control will insert the corresponding NC function before and after the marked area. **Further information:** "The Insert NC function window", Page 148
Block scan Program run button	When you press the **Block scan Program run** button, the control opens the current file in the **Program Run** operating mode, as well as the **Block scan** window for the currently selected NC block.

1.1.4 Technology-specific NC programming (#156 / #4-04-1)

Topic	Description
Cylindrical grinding with **FUNCTION MODE GRIND**	Use **FUNCTION MODE GRIND** (#156 / #4-04-1) to activate cylindrical grinding and select a kinematic model.
	Cylindrical grinding on a milling machine enables you to perform complete machining tasks on a machine without rechucking. Cylindrical grinding helps you to attain higher accuracies and better surface definitions than with turning. **Further information:** "Switching the operating mode with FUNCTION MODE", Page 154

1.1.5 Path functions

Topic	Description
Programming a straight line **L** with **actual position capture**	Press the **actual position capture** key to program a straight line **L** with the actual positions of all defined axes. The control inserts the straight line **L** after the selected NC block. **Further information:** "Straight line L", Page 224

1.1.6 Programming techniques

Topic	Description
Control structures (e.g., with **IF** or **ELSE**)	The control provides the NC functions for programming the control structures. The control provides the following NC functions: **IF**, **ELSE IF** and **ELSE** case analyses **FOR** and **WHILE** program loops - Expanded control of **BREAK** and **CONTINUE** program loops Using control structures, you can program the NC program more clearly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends. **Further information:** "Control structures", Page 289

1.1.7 Files

Topic	Description
Navigation path	In the file management, you can open a history of up to 20 previously used paths. **Further information:** "Areas of file management", Page 445
	You can edit the current navigation path.
User-defined filters	In the file management, you can define a user-defined filter for any desired file types. The filter is saved until you overwrite it. **Further information:** "Creating or changing a user-defined filter", Page 450
The **Open File** workspace	If you have opened the **Open File** workspace in **Editor** operating mode and only tables are selected, the control displays the **Use in simulation** button. **Further information:** "The Simulation settings window", Page 768
	The **Open File** workspace opens a file preview that you can show or hide. **Further information:** "The Open File workspace", Page 452

1.1.8 Text editor

Topic	Description
The **Text editor** workspace	Editing functions (e.g., tab stops) have been added to the **Text editor** workspace.
	Further information: "The Text editor workspace", Page 459
	The **Text editor** workspace contains **NC editor settings** (e.g., to replace tab stops with spaces).
	In the **Text editor** workspace, you can open and edit any desired file types.

1.1.9 Collision monitoring

Topic	Description
Set up fixtures (#140 / #5-03-2)	If the active unit of measure is inch, the control converts mm to inches within the **Set up fixtures** function.
Advanced checks	The **Advanced checks** function now includes the **Collision between workpiece and machine** check.
	The control displays a warning in case of collisions between the workpiece and the machine (e.g., spindle). The control does not consider the tool and the workpiece fixture.
	Further information: "Advanced checks in the simulation", Page 481

1.1.10 Multiple-axis machining

Topic	Description
Contour planing with **FUNCTION SHAPING** (#96 / #7-04-1)	Contour planing, also known as shaping, enables you to create sealing surfaces with a high surface definition, for example. When **FUNCTION SHAPING** is active, the control automatically moves the tool toward the contour during traverse movements. Using **FUNCTION SHAPING**, this automatic tracking also enables you to perform engraving, engine turning, or beveling.
	Further information: "Workpiece shaping with FUNCTION SHAPING (#96 / #7-04-1)", Page 527
Cylinder surface machining with **CYLINDER SURFACE** (#8 / #1-01-1)	The **CYLINDER SURFACE** NC function allows you to machine the cylinder surface with various NC functions, for example OCM cycles (#167 / #1-02-1), pocket milling cycles or path functions.
	Further information: "Cylinder surface machining with CYLINDER SURFACE (#8 / #1-01-1)", Page 510

1.1.11 Programming with variables

Topic	Description
Variable: named parameters	The control provides the named parameters variable type. The variable name of named parameters consists of a freely chosen designation enclosed in curly brackets (e.g., **{DEPTH_1}**). You can assign numerical and alphanumerical values to named parameters. **Further information:** "Variables: Q, QL, QR, QS parameters and named parameters", Page 599
Format strings	The control provides the **FMT** syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings. You can use formatted string parameters in the following NC functions: ■ **String formula** ■ **SQL SELECT** ■ **TEXT** within **FUNCTION REPORT** **Further information:** "Format strings", Page 649
Q parameter list window	The machine manufacturer can define language-sensitive descriptive texts for variables. In the **Q parameter list** window, you can select between the standard descriptive text and the machine manufacturer's text or enter a descriptive text. **Further information:** "The Q parameter list window", Page 603
FN 18: SYSREAD (ISO: **D18**)	The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced: ■ **FN 18: SYSREAD (D18) ID71 NR20**: machining information for dressing (#156 / #4-04-1) 　■ **IDX17**: wheel side to be dressed 　■ **IDX18**: grinding tool type 　■ **IDX19**: number of the active dressing cycle ■ **FN 18: SYSREAD (D18) ID720 NR0 IDX1**: status of a reciprocating movement during grinding (#156 / #4-04-1)

1.1.12 The Contour graphics workspace

Topic	Description
Auto draw	In **Editor** mode, the control provides the **Auto draw** toggle switch. If you select an NC block and activate the toggle switch, the control will show a real-time graphic of the subsequently programmed contour in the **Contour graphics** workspace. If you mark several NC blocks and activate the toggle switch, the control will draw the contour of the marked NC blocks. **Further information:** "Auto draw", Page 684

1.1.13 ISO

Topic	Description
NC function **G79|G00**	The control calls the most recently programmed machining cycle at the position you defined in the NC block with **G79|G00**. The control moves to the defined position at rapid traverse. **G79|G00** corresponds to the Klartext syntax **CYCL CALL POS** with **FMAX**. **Further information:** "Cycle call", Page 712
Syntax search	If the **ISO editor** toggle switch is active, you can search for identical syntax elements in different NC blocks.

1.1.14 User aids

Topic	Description
Issuing notifications with **FUNCTION REPORT**	With the **FUNCTION REPORT** NC function, the controls issues notifications under program control. You can define the notification text yourself. If the machine manufacturer or another provider has saved notifications as a PO file, you can also output these notifications. **Further information:** "Issuing messages with FUNCTION REPORT", Page 759
Search column in the **Text editor** workspace	In the **Text editor** workspace, the control provides the **Search** column. The search works in the same way as in the **Program** workspace. **Further information:** "The Search column in the Program and Text editor workspaces", Page 741
Context menu in the **Document** workspace	In the **Document** workspace, the control provides a context menu with additional functions for every file type (e.g., navigating back within the opened files). **Further information:** "Context menu in the Document workspace", Page 752
Showing NC blocks	You can show hidden NC blocks with the **BACKSPACE** key. **Further information:** "Hiding NC blocks", Page 735
Calculator	The calculator's keyboard provides the following input options: ■ The **P** key corresponds to **PI** ■ The **RETURN** or **ENT** keys correspond to **=** ■ The **DEL** key corresponds to **DEL** **Further information:** "Calculator", Page 754
	When the **actual position capture** key is pressed, the control shows the current axis positions in the calculator. You can copy the current value of an axis into the calculator.
GOTO function in the **Text editor** workspace	In the **Text editor** workspace, use the **GOTO record** button to define the number of a line to be selected by the control. **Further information:** "GOTO function", Page 732

1.1.15   The Simulation **workspace**

Topic	Description
The **Workpiece options** column	The control provides the following functions also in the **Manual** and **Program Run** operating modes: ■ **Reset the workpiece** ■ **Remove the chips**
The **Simulation settings** window	The **Simulation settings** window is available in the **Program Run** and **Manual** operating modes. You can select whether the control will show a solid-model view.
	In the **Tables** area, the control shows the **Reset** button. With the **Reset** button, the control selects the same tables for the simulation that are active for program run. **Further information:** "The Simulation settings window", Page 768
Overview window with active simulation	If the simulation of another NC program is currently running, the control shows a window with the name of this NC program above the function bar. If you double-tap or click this window, the control switches from the active tab to the NC program currently being simulated. **Further information:** "The Simulation workspace", Page 761

1.1.16   Tables

Topic	Description
The **Table** workspace	In the header of the **Table** workspace, the control may show the **"Rules not met" filter** icon, if applicable.
	The control displays only the rows that do not meet the rules defined by the machine manufacturer in **CfgTableCellCheck** (no. 141300). **Further information:** "Icons and shortcuts", Page 814
Create new table window	In the **Create new table** window, you can choose between mm and inches as the unit of measure. **Further information:** "The Create new table window", Page 811

1.2 Modified or extended functions

1.2.1 Programming fundamentals

Topic	Description
Marking an area using identical syntax elements	If you press **SHIFT + UP** or **SHIFT + DOWN** during editing, the control will jump to the next NC block with the same syntax element. In doing so, the control marks the two NC blocks and the area in between.
	Further information: "Searching for identical syntax elements in different NC blocks", Page 146
Text editor mode	The **Autocomplete in text mode** function additionally offers the possibility of choosing a tool via the selection dialog, for example.
	Further information: "Text mode", Page 150
	Using the arrow keys to the left and right, you can transfer syntax elements from the selection menu of the autocomplete feature to the NC program.

1.2.2 Technology-specific NC programming (#156 / #4-04-1)

Topic	Description
Dressing of grinding tools	You can dress grinding tools with assigned tool carrier kinematics.

1.2.3 Tools

Topic	Description
Preselecting a tool with **TOOL DEF**	You can no longer program **L** and **R** in the **TOOL DEF** NC function. The control issues an error message during execution.
	Further information: "Tool pre-selection using TOOL DEF", Page 213

1.2.4 Programming techniques

Topic	Description
NC sequences	You can save up to 2000 sequential NC blocks as one NC component.
	Further information: "NC sequences for reuse", Page 305
	You can save user-defined folder icons for the subfolders of the NC components.
CALL LBL	In the selection menu of **CALL LBL**, the control shows the comment in addition to the number or name of the label.

1.2.5 Compensations

Topic	Description
FUNCTION TURNDATA CORR	**FUNCTION TURNDATA CORR** is available with the Adv. Spindle Interpol. software option (#96 / #7-04-1).
	With **FUNCTION TURNDATA CORR-WPL**, you can define a delta value in the X direction as a diameter value using the **DXL-DIAM:** syntax element.
	Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422

1.2.6 Files

Topic	Description
The **Update TAB / PGM** function	The control supports only tables with UTF-8 character encoding. With the **Update TAB / PGM** function, the control changes the character encoding to UTF-8, if applicable. **Further information:** "Adapting files", Page 456
Sequence of the file information	The control displays the file information in the sequence: date, time and file size. **Further information:** "Areas of file management", Page 445
The **Quick selection new table** workspace	In the **Active tables for simulation** area, you can open the tool tables selected in the **Simulation settings** window as tabs in the **Tables** operating mode. **Further information:** "Quick selection new table workspace", Page 453

1.2.7 Text editor

Topic	Description
The **Text editor** workspace	Pressing the **ENT** key inserts a line break in the **Text editor** workspace. **Further information:** "The Text editor workspace", Page 459
Note regarding external file modifications	If the currently opened file was modified by another editor, the control will update the file content and display a note.

1.2.8 Collision monitoring

Topic	Description
Set up fixtures (#140 / #5-03-2)	The contents and the representation of the error estimate diagram have been changed. The error estimate diagram shows for each touch point the distance of the touch point from the nominal position of the 3D model. The columns of the diagram are transparent until the status of all axes is green.
Combining fixtures	The individual components of a combined fixture retain assigned attributes (e.g., colors).
Advanced checks	The **Workpiece collision** check has been renamed to **Collision between workpiece and tool**. **Further information:** "Advanced checks in the simulation", Page 481

1.2.9 Programming with variables

Topic	Description
Q parameter list window	Via the **Global search** toggle switch you can choose whether the control searches all columns of the **Q parameter list** window or only the currently selected window.
	Further information: "The Q parameter list window", Page 603
FN 18: SYSREAD (ISO: **D18**)	If you read the data of the current tool with **FN 18: SYSREAD** (ISO: **D18**) (e.g., **ID950**), the control shows the data of the new tool right from the start of the tool change.
	Further information: "Read system data with FN 18: SYSREAD", Page 629

1.2.10 User aids

Topic	Description
The **Search** column	The search also considers space characters at the start of the search term.
	Further information: "The Search column in the Program and Text editor workspaces", Page 741
The **Structure** column	The control shows ISO cycles in the **Structure** column.
	Further information: "The Structure column in the Program workspace", Page 737
Selecting text with touch operation	When selecting text during touch operation, the control will show two selection symbols below the text. With these symbols you can change the marked area by dragging.

1.2.11 Pallet machining and job lists

Topic	Description
Editing the pallet table	Although a pallet table has been selected in **Program Run** mode of operation, you are able to edit in **Editor** mode.

1.2.12   Tables

Topic	Description
Table filters	The control displays user-defined filters under the **All** filter. You can select and deselect the user-defined filters.
	When you tap or click a filter once, the control activates only the selected filter in the corresponding area.
	When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.
The **Form** workspace	The control displays the contents of some tables in groups within the **Form** workspace. Contents that are not assigned to any group are displayed under **Not categorized**. In the **Tool_management** application, for example, the area contains tool parameters that are irrelevant to the current tool type.
Characteristics of the table cells	The machine manufacturer can define the color and font of table cells.
The **TABDATA** function	In the **TABDATA** functions, you can also enter the table row as a number or numerical parameter. **Further information:** "Accessing table values ", Page 823
The **Filter** column	For tables without default filters, the control displays the **Filter** column as soon as a user-defined filter is saved.
Machine manufacturer settings	In the optional machine parameter **choice** (no. 105704), the machine manufacturer can define toggle switches of the **Form** workspace. The machine manufacturer can change the icon and adapt the background color.
The **Pocket table** application	The **Reset row** button has been removed from the **Pocket table** application.

2

About the User's Manual

2.1   Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:
- Operating the machine
    - Setting up tools
    - Setting up workpieces
    - Machining workpieces
    - Eliminating possible errors during program run
- Creating and testing NC programs
    - Creating NC programs at the control or externally using a CAM system
    - Using the Simulation mode to test the NC programs
    - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:
- Basic technical understanding (e.g., spatial imagination and the ability to read technical drawings)
- Basic knowledge in the field of metal cutting (e.g., understanding the meaning of material-specific parameters)
- Safety instructions (e.g., understanding possible dangers and how to avoid them)
- Training on the machine (e.g., compreheding axis directions and the machine configuration)

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

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

2.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as a 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: 1358774-xx
    - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe cycles and machining cycles are not included.
    ID: 1358773-xx
    - The **Machining Cycles** User's Manual contains all functions of the machining cycles.
    ID: 1358775-xx
    - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
    ID: 1358777-xx
- As PDF files, sub-divided according to the printed versions or as the **Complete Edition** User's Manual, containing all modules
ID: 1369999-xx
TNCguide
- As an HTML file for use as the integrated product aid **TNCguide**: directly on the control
TNCguide

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended use", Page 58

Further information products for users

The following information products are available:

- The **overview of new and modified software functions** informs you about the innovations of specific software versions.
TNCguide
- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
    - Machine parameters of the **MPs for setters** application
    - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
    - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions
    TNCguide
- The **Functions of the TNC7** brochure informs you about the functions of the TNC7 in comparison with the TNC 640
ID: 1387017-xx.
HEIDENHAIN brochures
- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).
HEIDENHAIN brochures
- The **NC Solutions** database offers solutions for frequently occurring tasks.
HEIDENHAIN NC solutions

2.3 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.

2.4   Notes on using NC programs

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

Change the following contents as needed:

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

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

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

> **i**   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.

2.5 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 47

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 58

Related topics

- The **Help** workspace
  Further information: "The Help workspace", Page 726

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.

> **i** 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 language version (e.g., **en** for English).
>
> With the steps provided you can also update the **TNCguide** version. Updating may be required (e.g., after a software update).

Description of function

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

Further information: "The Help application", Page 52

Further information: "The Help workspace", Page 726

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

Further information: "Icons", Page 53

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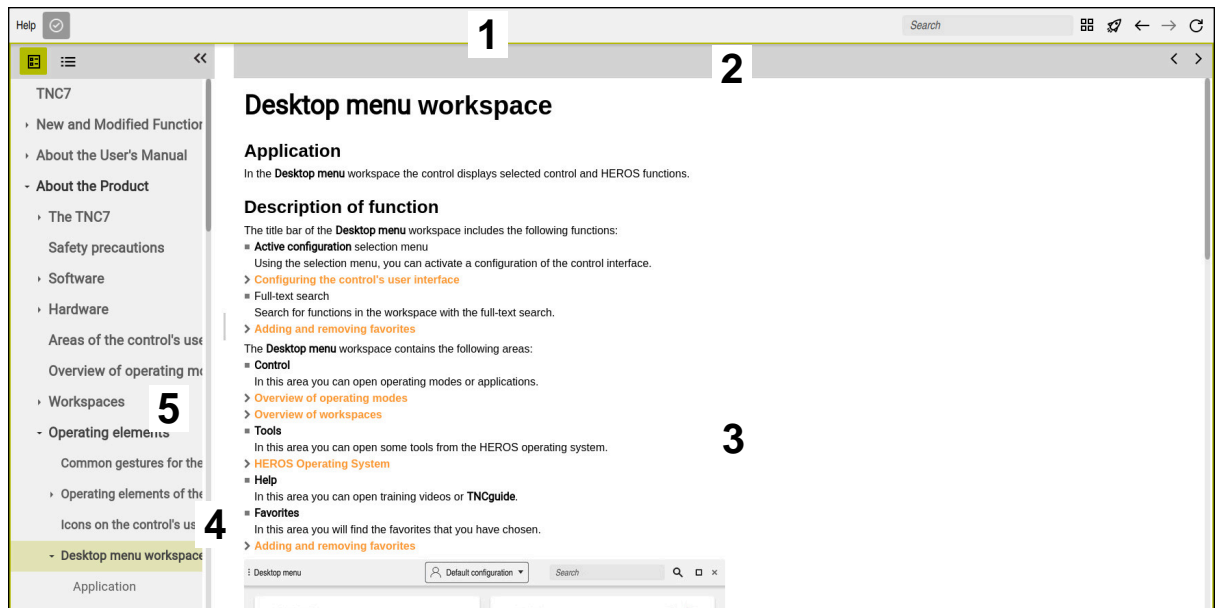
Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

1   Title bar of the **Help** workspace
    Further information: "The Help workspace", Page 53

2   Title bar of the integrated product aid **TNCguide**
    Further information: "TNCguide ", Page 53

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:** "Searching in TNCguide", Page 54
⊞	**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 55

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
⑦	**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 52

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 726

2.5.1 Searching 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 string in **Search**

> 🛈 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

2.5.2 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 50

▶ 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 761

2.6 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

3

About the Product

3.1   The TNC7

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

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

Functionality enhancements make it possible to go beyond milling and drilling in order to perform turning and grinding operations, for example,

Further information: "Technology-specific NC programming", Page 153

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.

3.1.1   Proper and intended use

The information about proper and intended use supports you in safely handling a product such as a machine tool.

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

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

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

> **ℹ**   Contribute actively to increasing the safety by reporting any missing or misleading information.
> **Further information:** "Contacting the editorial staff", Page 56

3.1.2 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.

3.2 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.

> ℹ 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 **Program Run** Single Block 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

3.3 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.

ℹ️ The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 63

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

ℹ️ 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
817620-19	TNC7
817621-19	TNC7 E
817625-19	TNC7 Programming Station

⚙️ Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine.

Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.

If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, the Adv. Function Set 2 software option (#9 / #4-01-1) is limited to 4-axis interpolation.

3.3.1 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 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 can be equipped with a **SIK** or **SIK2** plug-in board. Depending on which one is used, the numbers of the software options differ.

> **i** The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash (e.g., (#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. The control also shows whether is equipped with the **SIK** or **SIK2**.

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

Software option	Definition and application
Control Loop Qty. (#0-7 / #6-01-1*)	**Additional control loop** A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required, for example, for detachable and motor-driven tilting tables. If your control features a **SIK2**, you can order this software option multiple times and enable up to 24 control loops.
Adv. Function Set 1 (#8 / #1-01-1)	**Advanced functions (set 1)** On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: ▪ Tilting the working plane (e.g., with **PLANE SPATIAL**) **Further information:** "PLANE SPATIAL", Page 353 ▪ Programming of contours on a developed cylinder 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 569 ▪ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Adv. Function Set 2 (#9 / #4-01-1)	**Advanced functions (set 2)** On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces. The software option includes the following functions: ▪ **TCPM** (tool center point management): Automatic tracking of linear axes during rotary axis positioning **Further information:** "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393 ▪ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** "3D tool compensation (#9 / #4-01-1)", Page 424 ▪ Manual moving of axes in the active tool coordinate system **T-CS** ▪ Interpolation in up to six axes (max. four axes in case of an export version) The advanced functions (set 2) can be used to produce free-form surfaces.

Software option	Definition and application
HEIDENHAIN DNC (#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.
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
Global PGM Settings (#44 / #1-06-1)	**Global Program Settings (GPS)** This software option can be used for superimposed coordinate transformations and handwheel movements during program run without adapting the NC program. With GPS you can adapt externally created NC programs to the machine and increase flexibility during program run. **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:** Measuring Cycles for Workpieces and Tools User's Manual
Turning (#50 / #4-03-1)	**Mill-turning** This software option offers a comprehensive milling-specific function package for milling machines with rotary tables. The software option includes the following functions: ■ Turning-specific tools ■ Turning-specific cycles and contour elements such as undercuts ■ Automatic tool-tip radius compensation Mill-turning enables mill-turning machining operations on only one machine, thus reducing, for example, the setup work effort considerably. **Further information:** "Turning operations (#50 / #4-03-1)", Page 157
KinematicsComp (#52 / #2-04-1)	**KinematicsComp** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsComp, the control can correct position and component errors in three dimensions. This means it can spatially compensate for the errors of rotary and linear axes. Compared to KinematicsOpt (#48 / #2-01-1), the compensations are even far more comprehensive. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual
OPC UA NC Server (#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 the **SIK2**, you can order this software option multiple times and enable up to ten 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.** (#0-7 / #6-01-1*)", Page 64
8 Additional Axes (#78 / #6-01-1*)	**Eight additional control loops** **Further information:** "**Control Loop Qty.** (#0-7 / #6-01-1*)", Page 64

Software option	Definition and application
3D-ToolComp (#92 / #2-02-1)	**3D-ToolComp** only in connection with Advanced Function Set 2 (#9 / #4-01-1)
	With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated for using a correction value table.
	3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example.
	Further information: "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439
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.
Adv. Spindle Interpol. (#96 / #7-04-1)	**Interpolating spindle**
	This software option enables interpolation turning and contour planing, as the control couples the tool spindle with the linear axes.
	The software option includes the following functions:
	■ Turning-specific tools in the turning-tool table
	■ **FUNCTION SHAPING** for contour planing
	Further information: "Workpiece shaping with FUNCTION SHAPING (#96 / #7-04-1)", Page 527
	■ Cycle **291 COUPLG.TURNG.INTERP.** and Cycle **292 CONTOUR.TURNG.INTRP.** for interpolation turning
	■ **FUNCTION TURNDATA CORR** for compensation of turning tools in the NC program
	Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422
	The interpolating spindle enables you to execute a planing or turning operation also on machines without rotary table.
Spindle Synchronism (#131 / #7-02-1)	**Spindle synchronism**
	This software option synchronizes two or more spindles and thus enables, for example, the manufacture of gears by hobbing.
	The software option includes the following functions:
	■ Spindle synchronism for special machining operations (e.g., polygonal turning)
	■ Cycle **880 GEAR HOBBING** only in connection with mill-turning (#50 / #4-03-1)
	Further information: User's Manual for Machining Cycles

Software option	Definition and application
Remote Desk. 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
Collision Monitoring (#140 / #5-03-2)	**Dynamic Collision Monitoring DCM version 2** This software option includes all functions of the Collision Monitoring software option (#40 / #5-03-1). 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 479 **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**

Software option	Definition and application
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
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), 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 789
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.
Grinding (#156 / #4-04-1)	**Grinding operations** This software option offers a comprehensive grinding-specific function package for milling machines. The software option includes the following functions: ▪ Grinding-specific tools including dressing tools ▪ Cycles for jig grinding, cylindrical grinding and dressing Grinding enables complete machining operations on just one machine, thus considerably reducing setup work and increasing accuracy, for example. **Further information:** "Grinding operations (#156 / #4-04-1)", Page 171
Gear Cutting (#157 / #4-05-1)	**Gear manufacturing** This software option enables the manufacture of cylindrical gears or helical gears of any angle. The software option includes the following cycles: ▪ Cycle **285 DEFINE GEAR** to define the gear geometry ▪ Cycle **286 GEAR HOBBING** ▪ Cycle **287 GEAR SKIVING** Gear manufacturing expands the scope of functionality of milling machines with rotary tables even without mill-turning (#50 / #4-03-1). **Further information:** User's Manual for Machining Cycles

Software option	Definition and application
Turning v2 (#158 / #4-03-2)	**Mill-turning version 2** This software option includes all functions of the Turning software option (#50 / #4-03-1). In addition, this software option offers the following advanced turning functions: ■ Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** ■ Cycle **883 TURNING SIMULTANEOUS FINISHING** The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation. **Further information:** User's Manual for Machining Cycles
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 provides **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
Process Monitoring (#168 / #5-01-1)	**Process monitoring** Reference-based monitoring of the machining process The control uses this software option to monitor defined machining sections during program run. The control compares changes in conjunction with the tool spindle or the tool with the values of a reference machining operation. **Further information:** User's Manual for Setup and Program Run

3.3.2 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

3.4 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 62

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

The control must have at least 16 GB of RAM, as the control will otherwise display a warning.

3.4.1 Touchscreen and keyboard unit
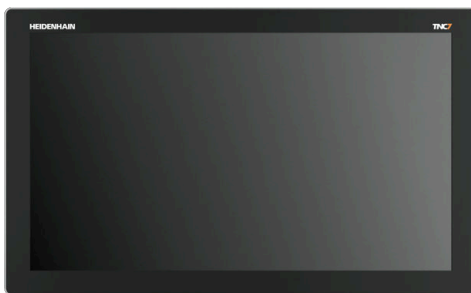
24" MC 366 with TE 361 (FS)   19" MC 356 with TE 350 (FS)
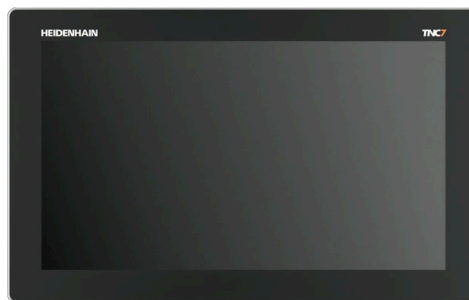
The TNC7 is available with various touchscreen sizes. Variants with 24" or 19" layout are available.

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 84

Further information: "Operating elements of the keyboard unit", Page 84

The machine operating panel is machine-dependent.

MB 350 (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

> ℹ Wear operating 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

> ℹ 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:
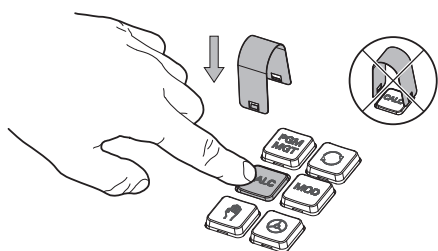
▶ Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

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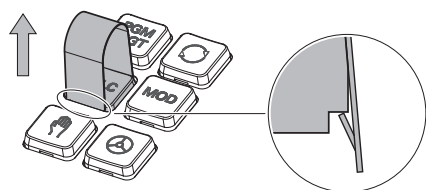
▶ Pull off the keycap
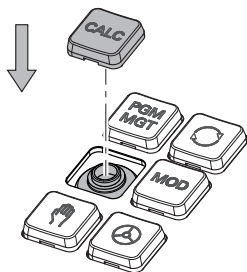
▶ Place the keycap onto the seal and push it down

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

▶ Verify proper seating and correct functionality

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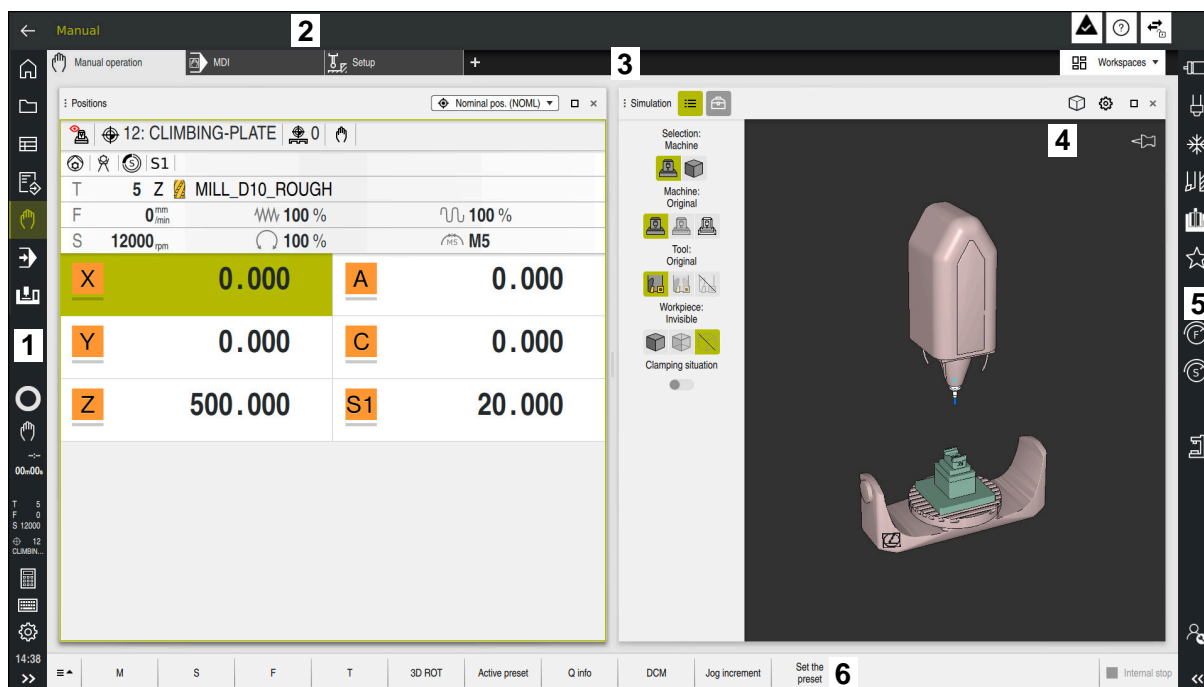
User interface of the control in the **Manual operation** application

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 78
   ■   Status overview
      Further information: User's Manual for Setup and Program Run
   ■   Calculator
      Further information: "Calculator", Page 754
   ■   Screen keyboard
      Further information: "Virtual keyboard of the control bar", Page 729
   ■   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 the **Dark Mode** function is available for selection.
      ■   **Font size**
   ■   Date and time

2   Information bar
- Active operating mode
- Message menu
- Symbols

       Further information: "Icons on the control's user interface", Page 93

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 80

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.

3.6 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 726
	▪ 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 442
⊞	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 808
⧉	In the **Editor** operating mode you can do the following:	Page 134
	▪ Create, edit and simulate NC programs	
	▪ Create and edit contours	
	▪ Create and edit pallet tables	
✋	The **Manual** operating mode contains the following applications:	
	▪ The **Manual operation** application	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	See the User's Manual for Setup and Program Run
	You can move the tool away from the workpiece, for example after a power failure.	
⇥	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.	See the User's Manual for Setup and Program Run
	You also execute pallet tables in this operating mode.	
⊠	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode.	See the User's Manual for Setup and Program Run
	Refer to your machine manual.	

Icon	Operating modes	Further information
	In the **Machine** operating mode the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	

3.7 Workspaces

3.7.1 Operating elements within the workspaces
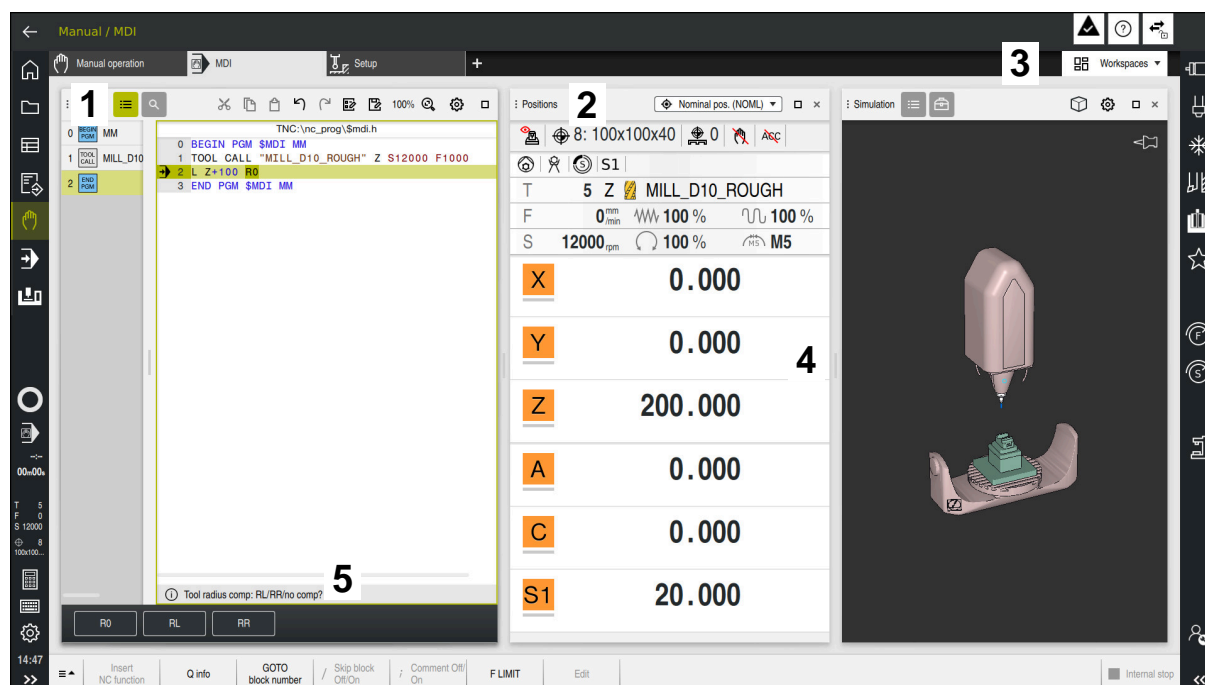
The control in the **MDI** application with three open workspaces

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.

3.7.2 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.

3.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function In the **Probing function** workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools or 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 789
Open File In the **Open File** workspace you select or create files, for example.	Page 452
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 442
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 454
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 820
The **Form** for pallets In the **Form** workspace the control shows the contents of the pallet table for the selected row.	Page 797

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
GPS (#44 / #1-06-1) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	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 96
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 726
Contour graphics In the **Contour graphics** workspace, the control can draw contours directly during programming. You can also use graphical programming by drawing contours and exporting them as NC blocks. In addition, you can import contours from existing NC programs and edit them graphically.	Page 675
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 136
Process Monitoring (#168 / #5-01-1) In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate up to six concurrent monitoring tasks for the corresponding monitoring sections. If required, monitoring tasks can be parameterized, replaced or removed.	See the User's Manual for Setup and Program Run
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 452

Workspace	Further information
Simulation In the **Simulation** workspace, the control shows the simulated or current movements, depending on the operating mode.	Page 761
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
Start/Login In the **Start/Login** workspace, the control shows the steps that are performed during startup.	Page 101
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. You can search in all tables and filter the table content.	Page 813
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 729
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

3.8   Operating elements

3.8.1   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:

Icon	Gesture	Meaning
●	Tap	Select element
◉	Double tap	■ Open an element (e.g., window in the **Settings** application) ■ Edit an NC block ■ Reset the graphic or 3D model to its original size
●	Long press	Open context menu ⓘ ■ If you are working with a mouse, click with the right mouse key. ■ If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds.
↕◉↔	Swipe	■ Scroll ■ Rotate the graphic or 3D model
↕◉↔	Drag	■ Change the selected area ■ Shift elements
↕◉◉↔	Two-finger drag	■ Move a graphic or 3D model ■ Shift drawing view in the **Contour graphics** workspace
◉◉↗	Spread	■ Change font size ■ Enlarge a graphic or 3D model
◉◉	Pinch	■ Reduce font size ■ Reduce a graphic or 3D model

3.8.2   Operating elements of the keyboard unit

Application

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

Further information: "Common gestures for the touchscreen", Page 84

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.

> ℹ️ 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 729

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 603 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
⬡X	Show the hidden NC block **Further information:** "Hiding or showing NC blocks", Page 735
? /	Hiding or showing NC blocks
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
+ = − * 8 ? / (9) 0	Performing calculations for numerical values in input fields and table cells

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 452
🔘	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 754
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 51

Operating modes

ⓘ On the TNC7 the operating modes of the control are allocated differently than on the TNC 640. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but, 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 134
⊐▷	While the NC program is running, open the **Simulation** workspace in the **Editor** operating mode **Further information:** "The Simulation workspace", Page 761

Keycaps for NC dialog

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

Key	Meaning
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function **Further information:** "Fundamentals of approach and departure functions", Page 253
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode **Further information:** "The Contour graphics workspace ", Page 675
CHF	Program a chamfer **Further information:** "Chamfer CHF", Page 226
L	Program a straight line segment **Further information:** "Straight line L", Page 224
CR	Program a circular arc with radius entry **Further information:** "Circular path CR", Page 232
RND	Program a rounding arc **Further information:** "Rounding RND", Page 227
CT	Program a circular arc with tangential connection to the preceding contour element **Further information:** "Circular path CT", Page 235
CC	Program a circle center or pole **Further information:** "Circle center point CC", Page 229
C	Program a circular arc with reference to the circle center **Further information:** "Circular path C ", Page 230
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** Measuring Cycles for Workpieces and Tools User's Manual
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 282
LBL CALL	Program a subprogram or a program section repeat **Further information:** "Calling a label with CALL LBL", Page 283

Key	Meaning
STOP	Program an intentional stop **Further information:** "Programming the STOP function", Page 554
TOOL DEF	Pre-select a tool in the NC program **Further information:** "Tool pre-selection using TOOL DEF", Page 213
TOOL CALL	Call the tool in the NC program **Further information:** "Using TOOL CALL to call a tool", Page 205
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)
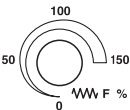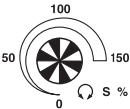
Keycaps for axis input and value input

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
⟨X⟩	Delete values during entry
+	Open position display of the status overview to copy axis values **Further information:** "Editing NC functions", Page 130 In the **Editor** operating mode and the **MDI** application, program a line **L** with the actual positions of all defined axes. **Further information:** "Straight line L with the values of the current position", Page 225
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 □	■ Directly position the cursor by using the number of an NC block, table row or machine parameter ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one page up in an NC program or table
PG DN	Go one page down in an NC program or table
📄	Mark the active application in order to navigate between applications
	Navigate between areas of an application

Potentiometers

Poten-tiometer	Function
(feed rate dial)	Increase or reduce the feed rate **Further information:** "Feed rate F", Page 211
(spindle speed dial)	Increase or reduce the spindle speed **Further information:** "Spindle speed S", Page 210

3.8.3   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

3.8.4 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 **Calculator**
▦	Open or close **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
✣	Activate or deactivate **Remember position** The control remembers the position of the window until it is shut down.

Icon or shortcut	Meaning
‹|›	Activate or deactivate **Change column width** You can change the width of the currently selected column.
⁘	**Scale** Resize windows
. . .	File functions are available
⭐	■ 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**
⑦	Call context-sensitive help **Further information:** "Context-sensitive help", Page 54
🔧	Dynamic collision monitoring (DCM) is deactivated **Further information:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 468

Icon or shortcut	Meaning
⇄🔒	**Secure connection configuration**
	External access to the control is active; all connections are using a secure connection configuration.
⇄	**Non-secure connection configuration**
	External access to the control is active and at least one connection uses a non-secure connection configuration.
🕐	Automatic program start active
	Further information: User's Manual for Setup and Program Run
▤	**Window manager**
	Select active applications in the background (e.g., HEROS functions window)
	ITC is connected and **Enabling VNC focus** is set
	Further information: User's Manual for Setup and Program Run
	Both the control and the ITC show the following icons.
✓	The focus is on the currently used device
	You can use the mouse and keyboard for the local and the remote device.
✗	The focus is on the remote device
	You cannot use the mouse and keyboard on the local device.
?	No focus assigned
	Inputs with the mouse and keyboard are locked until the focus has been assigned to the local or remote device.
Safety self-test	Self-test of the control is running

3.8.5 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 97

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 78

  Further information: "Overview of workspaces", Page 81

- **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 51

- **Favorites**

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

  Further information: "Adding and removing favorites", Page 97
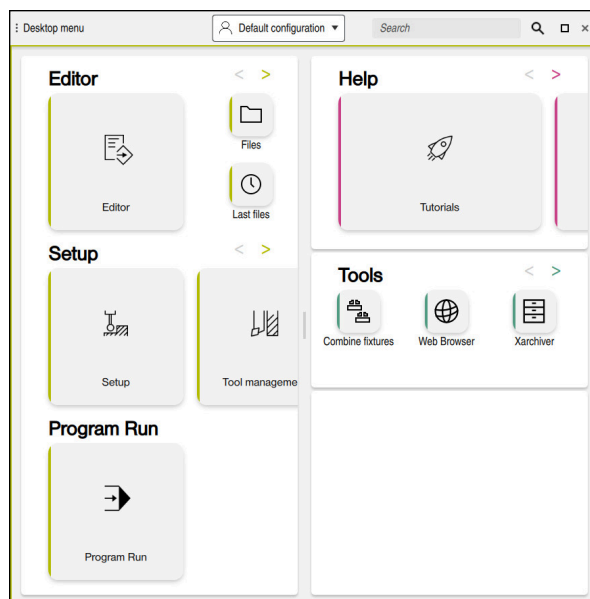
The **Desktop menu** workspace

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.

> ℹ️ 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
▶ Long-press 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:

▶ Long-press 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.

4

First Steps

4.1 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 on the machine and the control
- Programming and simulating a workpiece
- Switching the machine off

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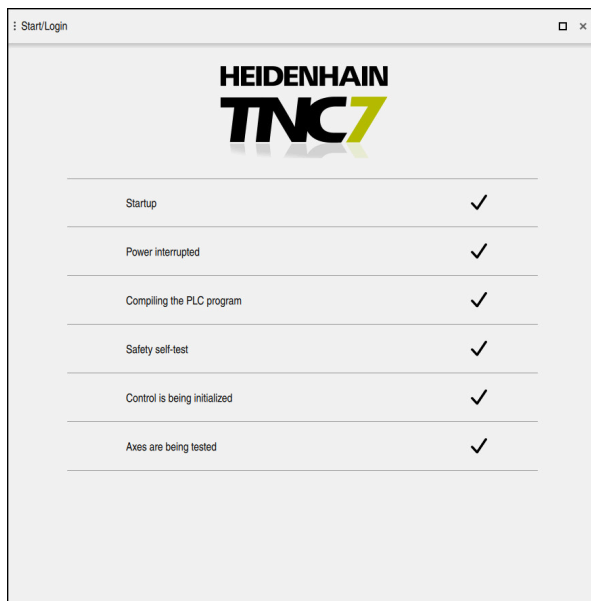
The **Start/Login** workspace

⚠ **DANGER**

Caution: hazard to the user!

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

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

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

To switch the machine on:

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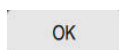
OK	▶ Press **OK**
	> The control compiles the PLC program.

▶ 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 125

4.3 Programming and simulating a workpiece

4.3.1 Example task 1339889

	Original drawing		Platte			Werkstoff: Material:	
	Scale	Format					
RoHS	1:1	A4	Plate				

Text:

Change No. C000941-05
Phase: Nicht-Serie

Einzelteilzeichnung / Component Drawing

●blanke Flächen/Blank surfaces

Maße in mm / Dimensions in mm

ID number

Werkstückkanten nach ISO 13715
Workpiece edges ISO 13715
-0.3 +0.3

Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2
General tolerances ISO 2768-mH ≦6mm: ±0,2

Tolerierung nach ISO 8015
Tolerances as per ISO 8015

Oberflächen nach ISO 1302
Surfaces as per ISO 1302

Oberflächenbehandlung:
Surface treatment:

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

HEIDENHAIN
DR. JOHANNES HEIDENHAIN GmbH
83301 Traunreut, Germany

Created	Responsible	Released		Version	Revision	Sheet	Page
M-TS			**D1339889-00-A-01**				1 of 1
11.01.2021			Document number				

4.3.2 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 134

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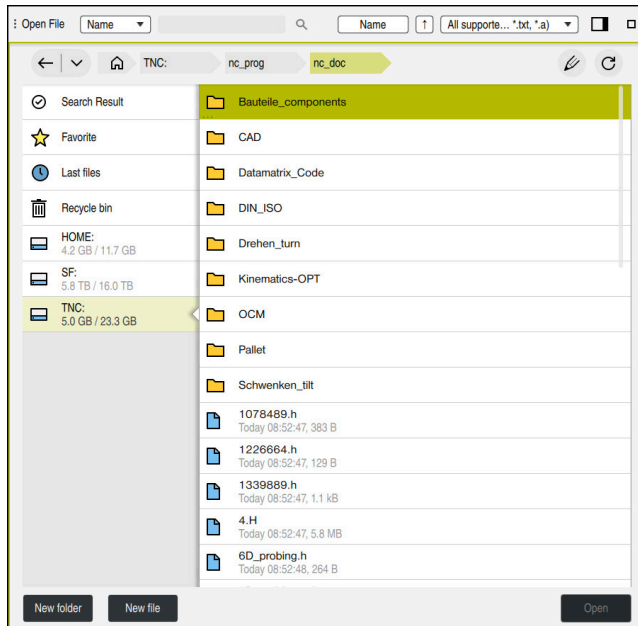
The **Open File** workspace in the **Editor** operating mode

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

▶ Select **Add**
> The control shows the **Quick selection** and **Open File** workspaces.

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

▶ Select a folder
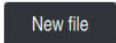
▶ Select **New file**

▶ Enter a file name (e.g., 1339899.h)
▶ Confirm with the **ENT** key
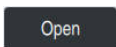
▶ Select **Open**
> The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

More detailed information
- The **Open File** workspace
  Further information: User's Manual for Setup and Program Run
- The **Editor** operating mode
  Further information: "The Editor operating mode", Page 134

4.3.4 Configuring the control's user interface for programming

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

> **i** 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: "Possible methods for editing", Page 128

- The **Form** column

   Further information: "The Form column in the Program workspace", Page 147

4.3.5 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.

> **i** 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.
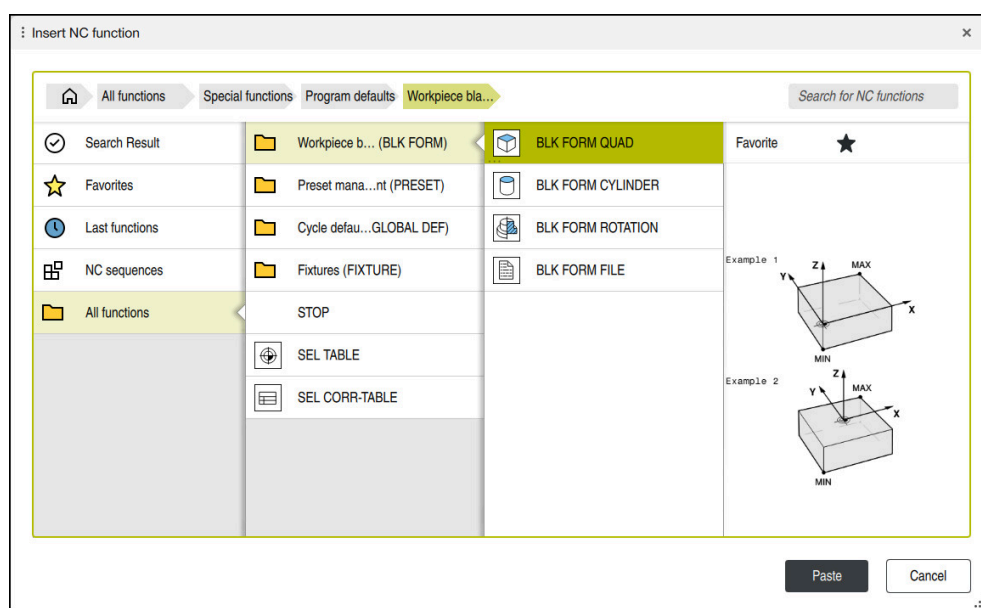
The **Insert NC function** window for workpiece blank definition

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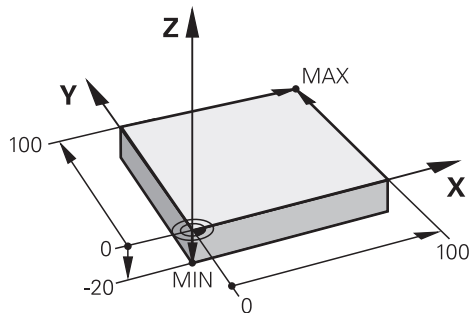
Cuboid workpiece blank with minimum point and maximum point

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

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

To define a cuboid workpiece blank:

| | ▶ Select **BLK FORM QUAD** |

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

| | ▶ Open the **Form** column |
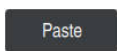
	▶ Select the tool axis (e.g., **Z**)
	▶ Confirm your input
	▶ Enter the smallest X coordinate (e.g., **0**)
	▶ Confirm your input
	▶ Enter the smallest Y coordinate (e.g., **0**)
	▶ Confirm your input
	▶ Enter the smallest Z coordinate (e.g., **-20**)
	▶ Confirm your input
	▶ Enter the largest X coordinate (e.g., **100**)
	▶ Confirm your input
	▶ Enter the largest Y coordinate (e.g., **100**)
	▶ Confirm your input
	▶ Enter the largest Z coordinate (e.g., **0**)
	▶ Confirm your input
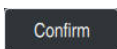
| Confirm | ▶ Select **Confirm** |
| | > The control concludes the NC block. |

The **Form** column with the defined columns

| 0 BEGIN PGM 1339889 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 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 186
- Reference points in the machine
  Further information: "Presets in the machine", Page 126

4.3.6 Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:
- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program

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

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

4.3.7 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:
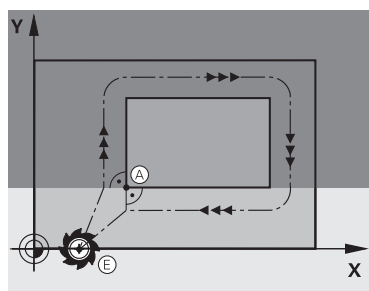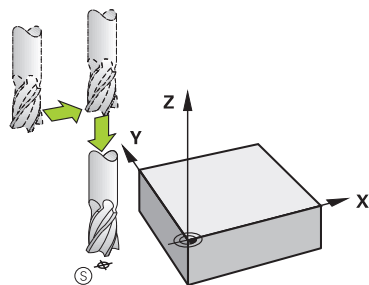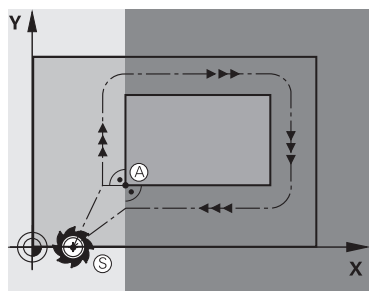
Help graphic	Position
	Starting point The following preconditions apply for the starting point: - No tool radius compensation - Approachable without danger of collision - Near to the first contour point The graphic shows the following information: If you define the starting point to be in the dark gray 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.
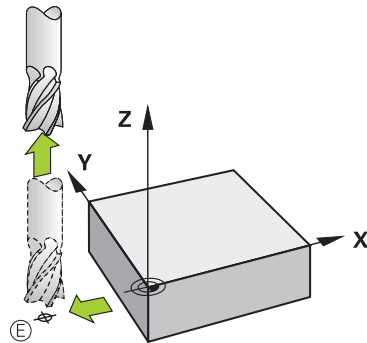
Help graphic	Position
	Departing from the end point in the tool axis Program the tool axis separately when departing from the end point.
	Identical starting and end points Do not program any tool radius compensation if the starting point and end point are the same. In order to make sure that the contour 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 253

4.3.8 Programming a simple contour
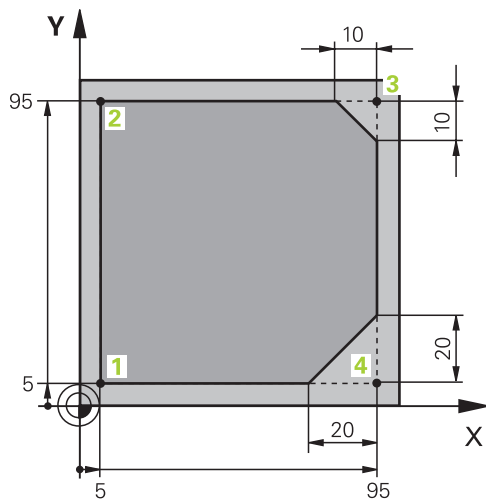
Workpiece to be programmed

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 106

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

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

Calling a tool

The **Form** column with the syntax elements of the tool call

To call a tool:

TOOL CALL	▶	Select **TOOL CALL**
	▶	Select **Number** in the form
	▶	Enter the tool number (e.g., **16**)
	▶	Select the tool axis **Z**
	▶	Select the spindle speed **S**
	▶	Enter the spindle speed (e.g., **6500**)
Confirm	▶	Select **Confirm**
	>	The control concludes the NC block.

3 TOOL CALL 16 Z S6500
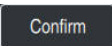
> The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).
> Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

Move the tool to a safe position

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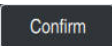
To move the tool to a safe position:

▶ Select the path function **L**

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

4 L Z+250 R0 FMAX M3

Pre-positioning in the working plane

To pre-position in the working plane:

▶ Select the path function **L**

▶ Select **X**
▶ Enter a value (e.g., **-20**)
▶ Select **Y**
▶ Enter a value (e.g., **-20**)
▶ Select the **FMAX** feed rate
▶ Select **Confirm**
> The control concludes the NC block.

5 L X-20 Y-20 FMAX

Pre-positioning in the tool axis

To pre-position in the tool axis:

▶ Select the path function **L**

▶ Select **Z**

▶ Enter a value (e.g., **-5**)

▶ Select the feed rate **F**

▶ Enter the value for the positioning feed rate (e.g., **3000**)

▶ If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)
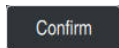
▶ Select **Confirm**

> The control concludes the NC block.

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

Approaching the contour
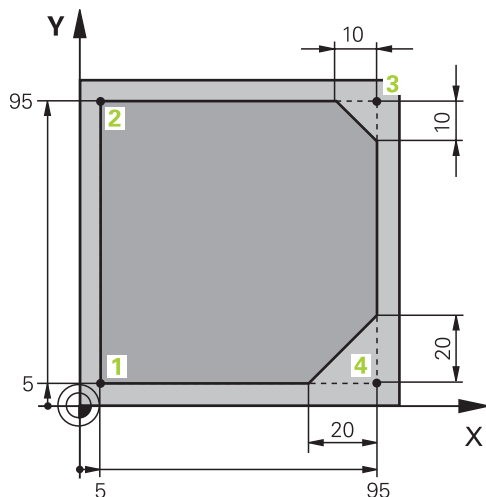
Workpiece to be programmed

The **Form** column with the syntax elements of an approach function

To approach the contour:

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

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

Confirm
- ▶ Select **Confirm**
- > The control concludes the NC block.
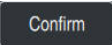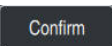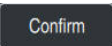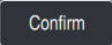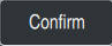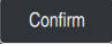
7 APPR CT X+5 Y+5 CCA90 R+8 RL F700

Machining a contour
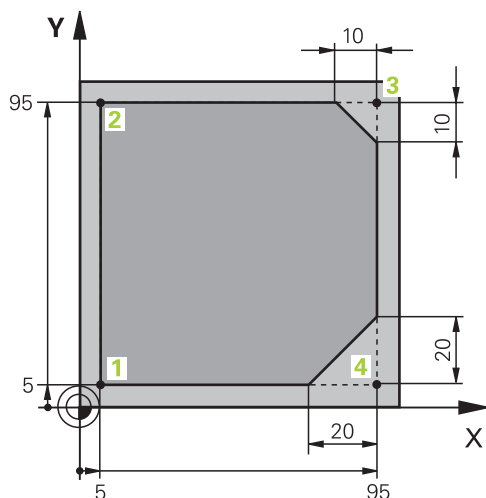
Workpiece to be programmed

To machine the contour:

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

▶ Select the path function **L**
▶ Enter the coordinates of contour point **3** that differ (e.g., **X 95**)
▶ Conclude the NC block with **Confirm**

▶ Select the path function **CHF**
▶ Enter the chamfer width (e.g., **10**)
▶ Conclude the NC block with **Confirm**

▶ Select the path function **L**
▶ Enter the coordinates of contour point **4** that differ (e.g., **Y 5**)
▶ Conclude the NC block with **Confirm**

▶ Select the path function **CHF**
▶ Enter the chamfer width (e.g., **20**)
▶ Conclude the NC block with **Confirm**

▶ Select the path function **L**
▶ Enter the coordinates of contour point **1** that differ (e.g., **X 5**)
▶ Conclude the NC block with **Confirm**

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

Departing from the contour

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

To depart from the contour:

APPR /DEP	▶ Select the **APPR DEP** path function
	> The control opens the **Insert NC function** window.
🖉	▶ Select **DEP**
🖉	▶ Select a departure function (e.g., **DEP CT**)
Paste	▶ Select **Paste**
	▶ For the center angle **CCA**, enter the departure angle (e.g., **90**)
	▶ Enter the departure radius (e.g., **8**)
	▶ Select the feed rate **F**
	▶ Enter the value for the positioning feed rate (e.g., **3000**)
	▶ If needed, enter a miscellaneous function **M**, such as **M9** (turn coolant off)
Confirm	▶ Select **Confirm**
	> The control concludes the NC block.

14 DEP CT CCA90 R+8 F3000 M9

Moving the tool to a safe position 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 miscellaneous function **M** (e.g., **M30**, end of program run)
▶ Select **Confirm**
> The control concludes the NC block and the NC program.
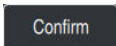
```
15  L  Z+250 R0 FMAX M30
```

More detailed information

■ Tool call

Further information: "Using TOOL CALL to call a tool", Page 205

■ Line **L**

Further information: "Straight line L", Page 224

■ Designation of the axes and the working plane

Further information: "Designation of the axes of milling machines", Page 124

■ Functions for approaching and departing from the contour

Further information: "Fundamentals of approach and departure functions", Page 253

■ Chamfer **CHF**

Further information: "Chamfer CHF", Page 226

■ Miscellaneous functions

Further information: "Overview of miscellaneous functions", Page 555

4.3.9 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 761

4.3.10 Simulating an NC program

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

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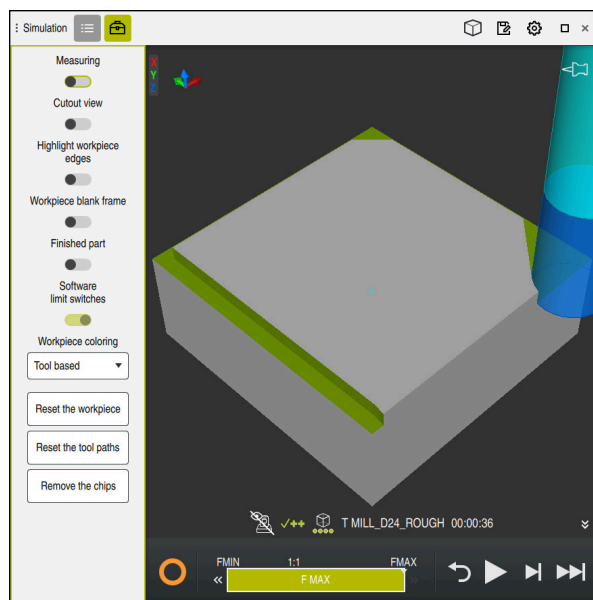
The **Simulation** workspace in the **Editor** operating mode

To start the simulation:

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

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

Definition

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

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

More detailed information

- The **Simulation** workspace
  Further information: "The Simulation workspace", Page 761

4.4 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

5

NC and Programming Fundamentals

5.1 NC fundamentals

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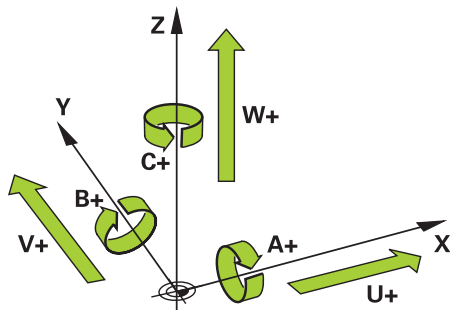
The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

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.

5.1.2 Designation of the axes of milling machines

The axes **X**, **Y** and **Z** on your machine are also 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.

5.1.3 Position encoders and reference marks

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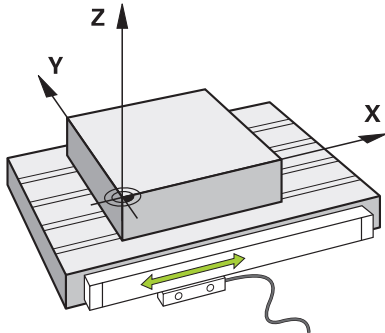
The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders. Rotary tables and rotary axes feature angle encoders.

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 310

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
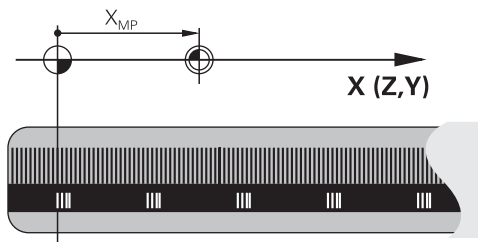
5.1.4 Presets in the machine

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 199

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 312
	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 558
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 560
	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 125
	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 316
	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.

5.2 Programming possibilities

5.2.1 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 220
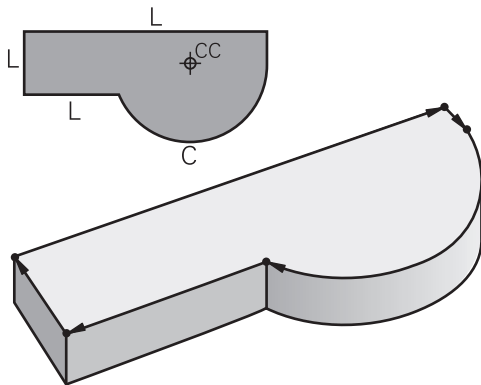
5.2.2 Graphical programming

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: "The Contour graphics workspace ", Page 675

5.2.3 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 553

5.2.4 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 282

5.2.5 Control structures

Using control structures, you can program the NC program more clearly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends.

Examples of control structures are case analyses and program loops.

You can program case analyses by using **IF** with a condition and **ELSE**, for example.

You can program loops by using **WHILE**, for example, and execute a program section several times.

Further information: "Control structures", Page 289

5.2.6 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 603

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, QS parameters and named parameters", Page 599

5.2.7 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.

Using a postprocessor, you can then generate the control- and machine-specific NC program in the CAM system. This results not only in programmable path functions but also **LN** lines with surface-normal vectors.

Further information: "Multiple-axis machining", Page 509

5.2.8 Possible methods for editing

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 have previously generated with a CAM system and transmitted to the control.

Related topics

- Using the **Program** workspace

  Further information: "Using the Program workspace", Page 144

- **Insert NC function** window

  Further information: "The Insert NC function window", Page 148

- Text mode

  Further information: "Text mode", Page 150

Description of function

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

> ℹ️ 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

  Frequently required NC functions such as path functions can be inserted via 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 729

- 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 148

- Inserting an NC function in Text mode

  In Text mode, the control offers automatic completion.

  Further information: "Text mode", Page 150

> ℹ️ If Text mode is active, then the **Klartext editor** toggle switch is located on the left and is dimmed.

Further information: "Inserting NC functions", Page 129

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 mode

  If the control can't automatically correct syntax errors in the NC program, it activates Text mode. You must correct all errors before you can switch to **Klartext editor** mode.

Further information: "Editing NC functions", Page 130

Inserting NC functions

Inserting an NC function directly with keys or buttons

To insert frequently needed NC functions:

▶ Select **L**

> The control creates a new NC block and starts the dialog.

▶ Follow the instructions in the dialog

Inserting an NC function by selecting it

To insert a new NC function:

Insert
NC function

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

Paste

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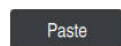
Inserting an NC function in Text mode

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 139
▶ 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

[icon]

▶ 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 Text mode

To edit an incorrect NC function in Text 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.

▶ Conclude the NC block

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

▶ Apply the proposal to the NC program with **Yes** or cancel auto-correction
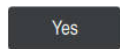
Inserting an axis value into an NC block

To enter the value of an axis into an NC block:

▶ Program the NC block up to the desired axis, such as **L X+10 Y...**

▶ Press the **actual position capture** key

> The control opens the position display of the status overview.

▶ Select the desired value of an axis

> The control inserts the value of the selected axis in the NC block.

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

■ In Text mode, the control may not be able to suggest a solution in every case.

■ 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 identical syntax elements in different NC blocks", Page 146

■ If you are editing an NC block and have not yet saved it, the **Undo** and **Redo** functions will affect the individual syntax elements of the NC function.

Further information: "Icons on the control's user interface", Page 93

■ 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.

5.3 Programming fundamentals

5.3.1 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 105
- NC programs using CAD files
  Further information: "CAM-generated NC programs", Page 538
- Structure of an NC program for contour machining
  Further information: "Structure of an NC program", Page 109

Description of function

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

Further information: "The Program workspace", Page 136

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

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

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

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

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

0 BEGIN PGM EXAMPLE MM	; Start of program
1 BLK FORM 0.1 Z X-50 Y-50 Z-20	
2 BLK FORM 0.2 X+50 Y+50 Z+0	; NC function for workpiece blank definition, consisting of two NC blocks
3 TOOL CALL 5 Z S3200 F300	; NC function for tool call
4 L Z+100 R0 FMAX M3	; NC function for straight-line traverse
* - ...	
11 M30	; NC function for ending 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 148
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 138
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 132

5.3.2   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:

- **Document**

  Further information: "The Document workspace", Page 454

- **Help**

  Further information: "The Help workspace", Page 726

- **Contour**

  Further information: "The Contour graphics workspace ", Page 675

- **Program**

  Further information: "The Program workspace", Page 136

- **Simulation**

  Further information: "The Simulation workspace", Page 761

- **Simulation status**

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

- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 729

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 789

Further information: "The Form workspace for pallets", Page 797

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

Further information: "The Job list workspace", Page 789

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:** "The Contour graphics workspace ", Page 675
	The control uses this icon to show that a pallet table is open. **Further information:** "Pallet machining and job lists", Page 787
	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 or **ISO editor**	If this toggle switch is set to active, then you are using dialog-guided programming. If this toggle switch is not set to active, then you are programming in Text mode. **Further information:** "Possible methods for editing", Page 128
Auto draw	If the toggle switch is set to active, then the control draws the contour that you are currently programming in the **Contour graphics** workspace. **Further information:** "Auto draw", Page 684
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Possible methods for editing", Page 128
GOTO block number or **GOTO record**	The control selects the block or row number that you defined. **GOTO block number** for **Program** workspace only **GOTO record** for **Text editor** workspace only **Further information:** "GOTO function", Page 732
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 603
/ Skip block Off/On	Hide NC blocks with the **/** character. 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 735
; 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 734
Edit	The control opens the context menu. **Further information:** "Context menu", Page 747
Block scan Program run	The control opens the file in the **Program Run** operating mode and opens the **Block scan** window for the currently selected NC block. This allows you to execute the NC program directly from the selected NC block. **Further information:** User's Manual for Setup and Program Run
Select in Program Run	The control opens the file in the **Program Run** operating mode and selects the first NC block. **Further information:** User's Manual for Setup and Program Run

Icon or button	Meaning
Start the simulation	The control opens the **Simulation** workspace and starts graphic simulation. **Further information:** "The Simulation workspace", Page 761

5.3.3 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
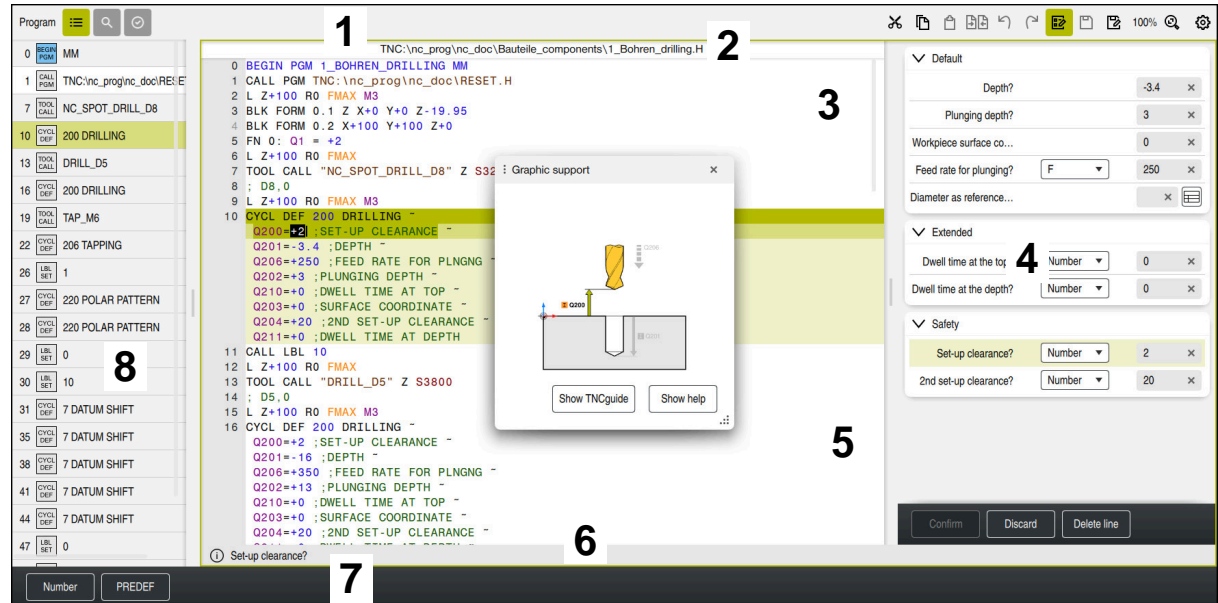
The **Program** workspace with active structure, help graphic, and form

1 Title bar

Further information: "Icons in the title bar", Page 138

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 138

4 The **Form** column

Further information: "The Form column in the Program workspace", Page 147

5 Help graphic of the syntax element being edited

Further information: "Help graphic", Page 139

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 737

Further information: "The Search column in the Program and Text editor workspaces", Page 741

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:

Icon or shortcut	Function
☰	Open and close the **Structure** column **Further information:** "The Structure column in the Program workspace", Page 737
🔍 **CTRL + F**	Open and close the **Search** column **Further information:** "The Search column in the Program and Text editor workspaces", Page 741
⊘	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 745
📝	Open and close the **Form** column **Further information:** "The Form column in the Program workspace", Page 147
100%	Font size of the NC program ℹ️ If you select the percent value, the control displays icons for increasing and decreasing the font size.
🔍⊕	Set font size of the NC program to 100%
⚙	Open the **Program settings** window **Further information:** "Settings in the Program workspace", Page 139

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.

Whether the control displays the help graphic depends on the setting **Show help graphics automatically** or the machine parameter **stdTNChelp** (no. 105405).

Further information: "Settings in the Program workspace", Page 139

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 51
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 726

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	The MDI application
Structure	✓	✓	✓
Edit	✓	-	✓
Klartext	✓	-	✓
Tables	-	✓	-
FN 16	-	✓	✓

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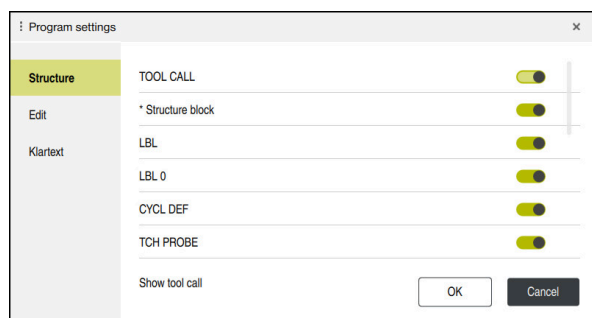
The **Structure** area in the **Program settings** window

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 737

The following structure elements are available:

- **TOOL CALL**
- *** Structure block**
- **LBL**
- **LBL 0**
- **CYCL DEF**
- **TCH PROBE**
- **ISO cycle**
- **MONITORING SECTION START** (#168 / #5-01-1)
- **MONITORING SECTION STOP** (#168 / #5-01-1)
- **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 - Press the **SPACE** soft key
	If the toggle switch is not active, you can open the selection menu by pressing **CTRL + SPACE**.
	Further information: "Text mode", Page 150
Allow syntax errors in text mode	If you activate the toggle switch, then the control can save NC blocks in Text mode, 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 130

Setting	Meaning
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 447
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. **ⓘ** If you select the **All Files (*.*)** filter, then 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 726
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.

Setting	Meaning
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 305
Hide NC functions that aren't available	If the toggle switch is active, the control will only display currently available NC functions in the **Insert NC function** window. If the toggle switch is not active, the control dims unavailable NC functions (e.g., for software options that are not enabled).
Put all path information in quotation marks	If the toggle switch is active, the control will automatically enclose path information in quotation marks when you select one of the following NC functions: ■ **CALL PGM** ■ Cycle **12 PGM CALL** ■ **FN 16 F-PRINT** ■ **FN 26 TABOPEN** The same setting is also available in the optional machine parameter **quotePaths** (no. 105414). The control will reconcile both setting options.
Display screen keyboard for editing	If a touchscreen is used, the control will display a context-sensitive virtual keyboard. A selection menu allows you to select the position of the virtual keyboard in the workspace or to hide the virtual keyboard.

Klartext area

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

The control offers the following settings as toggle switches:

Setting	Meaning
Skip comment	If you activate this toggle switch, the control skips the comment function during programming for all NC functions. **Further information:** "Adding comments", Page 734
Skip tool index	If you activate this toggle switch, the control skips the tool index for the following NC functions: ■ Calling a tool with **TOOL CALL** **Further information:** "Using TOOL CALL to call a tool", Page 205 ■ Preselecting a tool with **TOOL DEF** **Further information:** "Tool pre-selection using TOOL DEF", Page 213 **Further information:** User's Manual for Setup and Program Run

Setting	Meaning
Skip linear superimposed interpolated axis values	If you activate this toggle switch, the control skips the **LIN_** syntax element for the following NC functions: ■ Circular contour **C** **Further information:** "Circular path C ", Page 230 ■ Circular contour **CR** **Further information:** "Circular path CR", Page 232 ■ Circular contour **CT** **Further information:** "Circular path CT", Page 235 **Further information:** "Linear superimpositioning of a circular path", Page 237

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 832

■ **Tool correction**

Further information: "Compensation table *.tco", Page 842

■ **Workpiece correction**

Further information: "Compensation table *.wco", Page 844

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 622

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 or mark characters **Further information:** "Marking characters in an NC block", Page 146
	Long press	Open the context menu ℹ️ If you are working with a mouse, click with the right mouse key. **Further information:** "Context menu", Page 747
	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 750
	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 ■ Navigate within selection menus ■ During editing, search for the same syntax element in the NC program **Further information:** "Searching for identical syntax elements in different NC blocks", Page 146
▶　　　◀	■ Edit an NC block ■ During editing, navigate to previous or next syntax element ■ Arrow right: in text mode, take over a syntax element from the autocomplete feature
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 732 ■ 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	Enter a line break in text mode Insert a line break in the **Form** column for comments
ESC	Cancel editing without applying changes
Klartext editor	Select the **Klartext editor** mode or text mode **Further information:** "Editing NC functions", Page 130
Insert NC function	Open the **Insert NC function** window **Further information:** "Areas of the Insert NC function window", Page 148
Edit	Open the context menu **Further information:** "Context menu", Page 747

Searching for identical 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 selects 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.

> 🛈   ■ If you also keep the **SHIFT** key held down, the controls marks all NC blocks until the previous or next identical syntax element.
>   ■ You can search for identical syntax initiators in an NC program. Select the syntax initiator by double-tapping or double-clicking it.

Marking characters in an NC block

You can mark multiple characters within a single NC block.

How to mark multiple characters in a single NC block:

▶   Double-tap or double-click the NC block

>   The control marks the selected value.

▶   Use the "drag" gesture to expand or reduce the marked area

> 🛈   In Text mode you can mark any range of characters. In the **Klartext editor** mode you can mark only the characters of a value.
>   **Further information:** "Contents of an NC program", Page 132

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.
- When you open an NC program, the control checks whether the NC program is complete and syntactically correct.
- 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.
- If you mark characters through touch operation, the control displays two marker symbols below the cursor.
- Within the **Program** workspace, calculations aren't possible in entry fields for numerical values.

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 797
- Editing an NC function in the **Form** column
  Further information: "Editing NC functions", Page 130

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 a symbol next to the input area. The control uses this symbol to assign the next available number to the label.

5.3.4 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 305
- Inserting and editing NC functions

  Further information: "Possible methods for editing", Page 128

Description of function

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

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

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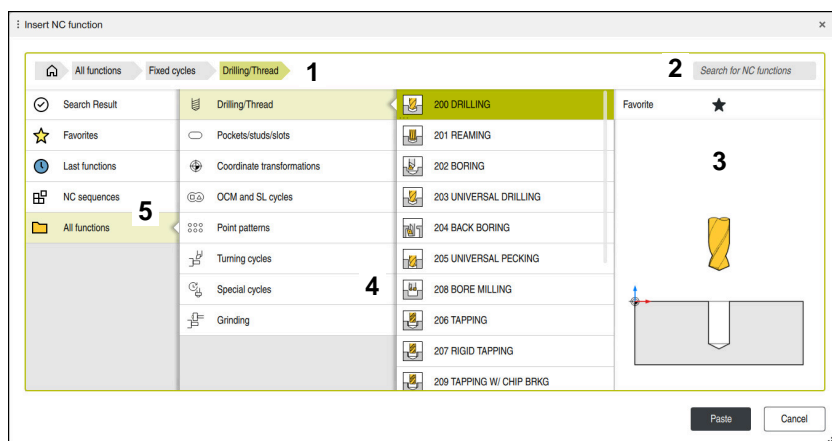
The **Insert NC function** window

1   Navigation path

    In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can edit the path or open a previous path from the History.

    Further information: "Areas of file management", Page 445

2   Searching

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

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

> **i** 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 offers the following possibilities for navigation:

 ■ **Search Result**

    The control shows the following search results:

 ■ NC functions or miscellaneous functions whose name includes the content being searched for (e.g., Cycle **4019** in a search for "19")

 ■ Equivalent or alternative NC functions (e.g., **PATTERN DEF** when searching for "pattern")

 ■ Replacement functions for older and partly obsolete 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 93

 ■ **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 305

 ■ **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 88

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 305

Notes

- For some NC functions, the **Insert NC function** window offers the possibility of inserting the start and end of the NC function into the NC program at the same time (e.g., **IF and END IF**).

  If you mark several NC blocks in the NC program and insert the combined NC functions, the control will insert the corresponding NC function before and after the marked area.

- 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.

5.3.5   Text mode

Application

The Text mode is a possibility for programming within the **Program** workspace. In Text mode you create and edit NC programs using the keyboard instead of the **Insert NC function** window.

Related topics

- Editing text files in the **Text editor** workspace

  Further information: "The Text editor workspace", Page 459

- Basics and operation of the **Program** workspace

  Further information: "The Program workspace", Page 136

Description of function

If the **Klartext editor** toggle switch in the function bar is set to inactive, then you are programming the NC programs in Text mode. Text mode works similar to a common text editor. For example, the arrow keys don't move the cursor from one syntax element to the next, but only to the next character.

Text mode supports all navigation possibilities of the **Program** workspace.

Further information: "Using the Program workspace", Page 144

If the control can't automatically correct syntax errors in the NC program, it activates Text mode.

Further information: "Editing NC functions", Page 130

In the **Program settings** window the control offers the following settings for Text mode:

- **Autocomplete in text mode**
- **Allow syntax errors in text mode**

Further information: "Settings in the Program workspace", Page 139

Auto-completion in Text mode

While programming in Text mode, the control opens a selection menu with all syntax elements that you can insert at the current cursor position.

If the **Autocomplete in text mode** toggle switch is set to active, the control displays the selection menu upon the following actions:

- Creating a new NC block
- Entering characters

  The control adapts the proposals to the entered characters.

- Pressing the **SPACE** key

If the toggle switch is not active, you can open the selection menu by pressing **CTRL + SPACE**.

You have the following options for entering the desired syntax element:

- Tap
- Click
- The right arrow key

  If you used the arrow keys to choose an element from the selection menu, or if there is only one element.

Further information: "Inserting NC functions", Page 129

Notes

- By default, the control displays only syntax elements without values when using auto-completion. If miscellaneous functions are possible, then the control displays all miscellaneous functions with numbers after the remaining possible syntax elements.

- If you are programming a cycle, the control offers the **Only downwardly-compatible cycle parameters** and **With optional cycle parameters** possibilities for auto-completion.

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

- If you press the right arrow key during auto-completion without clearly choosing an element, the control will close the selection menu.

- In the Text mode you can enter line breaks at any location. 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.

6

Technology-specific NC programming

6.1 Switching the operating mode with FUNCTION MODE

Application

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

Related topics

- Mill-turning operations (#50 / #4-03-1)

  Further information: "Turning operations (#50 / #4-03-1)", Page 157
- Grinding operations (#156 / #4-04-1)

  Further information: "Grinding operations (#156 / #4-04-1)", Page 171
- Editing kinematic models in the **Settings** application

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

Requirements

- Control adapted by the machine manufacturer

  The machine manufacturer defines which internal functions the control performs with this function. The machine manufacturer must define selection possibilities for the **FUNCTION MODE SET** function.
- For **FUNCTION MODE TURN**, the Turning software option (#50 / #4-03-1)
- For **FUNCTION MODE GRIND**, the Grinding software option (#156 / #4-04-1)

Description of function

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode.

With the **FUNCTION MODE MILL**, **FUNCTION MODE TURN** and **FUNCTION MODE GRIND** NC functions you activate a machine kinematics setting that the machine manufacturer defined and stored in the macro.

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

If turning mode is active, the control shows a corresponding symbol in the **Positions** workspace (#50 / #4-03-1).

If the cylindrical grinding mode is active, the control shows a corresponding symbol in the **Positions** (#156 / #4-04-1) workspace.

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

Input

11 FUNCTION MODE TURN "AC_TURN"	; Activate turning mode with the selected kinematic model

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Machining mode (MODE)

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION MODE	Syntax initiator for the machining mode
TURN, **MILL**, **GRIND** or **SET**	Select the machining mode or machine manufacturer setting
Name or **Parameter**	Name of a kinematic model or machine-manufacturer setting Text or string parameter Selection by means of a selection window Optional syntax element

Notes

> ## ⚠ WARNING
>
> **Caution: Danger to the operator and machine!**
>
> Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!
>
> - ▶ Clamp the workpiece in the spindle center
> - ▶ Clamp workpiece securely
> - ▶ Program low spindle speeds (increase as required)
> - ▶ Limit the spindle speed (increase as required)
> - ▶ Eliminate unbalance (calibrate)

> ## ⚠ WARNING
>
> **Caution: hazard to the user!**
>
> Very high physical forces are generated during cylindrical grinding, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!
>
> - ▶ Clamp the workpiece in the spindle center
> - ▶ Clamp workpiece securely
> - ▶ Program low spindle speeds (increase as required)
> - ▶ Limit the spindle speed (increase as required)
> - ▶ Eliminate unbalance (calibrate)

- In the optional machine parameter **CfgModeSelect** (no. 132200), the machine manufacturer defines the settings for the **FUNCTION MODE SET** function. If the machine manufacturer does not define the machine parameter, then **FUNCTION MODE SET** is not available.
- If the functions **Tilt working plane** (#8 / #1-01-1) or **TCPM** (#9 / #4-01-1) are active, you cannot select a different machining mode.
- The preset must be in the center of the turning spindle in turning mode.

6.2 Turning operations (#50 / #4-03-1)

6.2.1 Fundamentals

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

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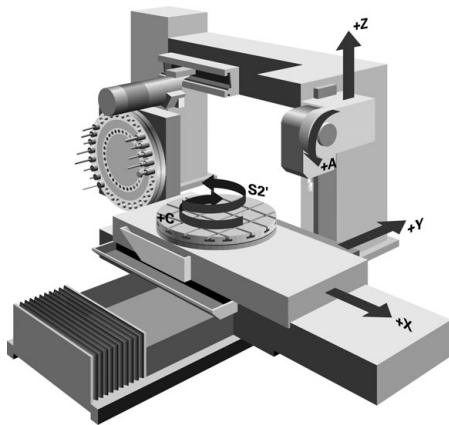
NC fundamentals for turning

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

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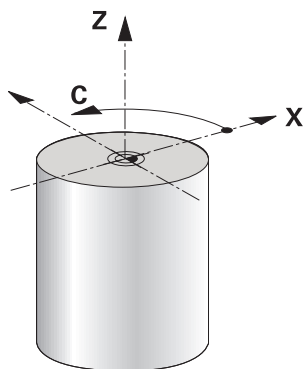
Workpiece preset for turning operations

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

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

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

Further information: "Using a facing head with FACING HEAD POS (#50 / #4-03-1)", Page 523

Production processes

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

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

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

Further information: User's Manual for Machining Cycles

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

Further information: "Inclined turning", Page 162

Tools for turning operations

When managing turning tools, other geometric descriptions than those for milling or drilling tools are required. The cutting-edge radius must be defined, for example, in order to apply cutter radius compensation. The control provides a special tool table for turning tools. In the **Form** workspace of the tool management, the control displays only the required parameters for the current tool type.

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

Further information: "Tool radius compensation (TRC) with lathe tools (#50 / #4-03-1)", Page 415

You can correct turning tool values in the NC program.

The control offers the following functions for this:

- Cutter radius compensation

  Further information: "Tool radius compensation (TRC) with lathe tools (#50 / #4-03-1)", Page 415

- Compensation tables

  Further information: "Tool compensation with compensation tables", Page 418

- The **FUNCTION TURNDATA CORR** function

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422

Notes

> ⚠️ **WARNING**
>
> **Caution: Danger to the operator and machine!**
>
> Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!
>
> ▶ Clamp the workpiece in the spindle center
> ▶ Clamp workpiece securely
> ▶ Program low spindle speeds (increase as required)
> ▶ Limit the spindle speed (increase as required)
> ▶ Eliminate unbalance (calibrate)

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

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

  Further information: "Overview of miscellaneous functions", Page 555

- During turning, the cutting edge and the center of the turning spindle must be at the same level. During turning, the tool therefore has to be pre-positioned to the Y coordinate of the turning-spindle center.

- In turning mode, diameter values are displayed on the X axis position display. The control then shows an additional diameter symbol.

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

- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).

- In turning mode, no coordinate conversion cycles are permitted except for the datum shift.

  Further information: "Datum shift with TRANS DATUM", Page 334

- In turning mode, the **SPA**, **SPB** and **SPC** transformations from the preset table are not permitted. If you activate one of these transformations, the control will display the **Transformation not possible** error message if executing the NC program in turning mode.

- The control does not use the **BLK FORM** function to generate the traverse paths for the turning cycles (#50 / #4-03-1). In this case, define **FUNCTION TURNDATA BLANK**.

  Further information: "Blank form update in turning with FUNCTION TURNDATA BLANK (#50 / #4-03-1)", Page 194

- The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

  Further information: "The Simulation workspace", Page 761

6.2.2 Technology values for turning operations

Defining the spindle speed for turning with FUNCTION TURNDATA SPIN

Application

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

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

Requirement

- Machine with at least two rotary axes, one of them as a rotary table axis
- Software option Turning (#50 / #4-03-1)

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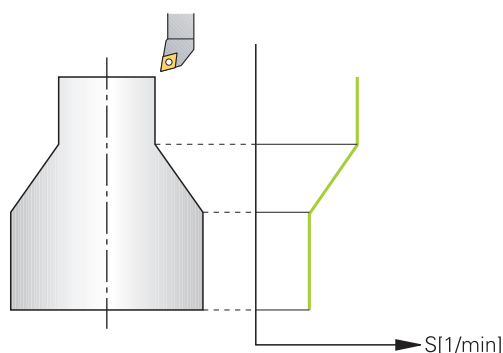
If you machine at constant cutting speed **VCONST:ON**, the control modifies the speed according to the distance of the tool tip to the center of the turning spindle. For positioning movements toward the center of rotation, the control increases the table speed; for movements away from the center of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

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

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

Input

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

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Turning functions ▶ Basic functions ▶ FUNCTION TURNDATA SPIN

The NC function includes the following syntax elements:

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

Notes

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

Feed rate

Application

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

Further information: "Interpreting the feed rate as mm/rev with M136", Page 581

Description of function

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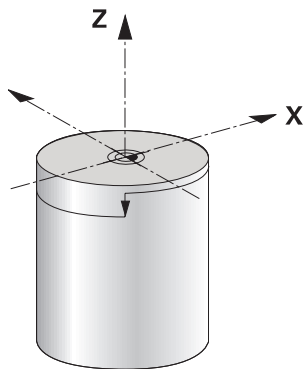
Note

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

6.2.3   Inclined turning

Application

In some cases, it might be necessary to bring rotary axes into a certain position in order to machine the workpiece as required. This can be necessary, for example, when you can only machine contour elements using a specific position due to tool geometry.

Requirement

■ Machine with at least two rotary axes, one of them as a rotary table axis
■ Software option Turning (#50 / #4-03-1)

Description of function
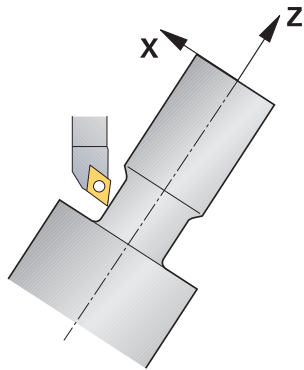
The control offers the following methods of inclined turning:

NC function	Description	Further information
M144	The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.	Page 586
M128	With **M128** the control behaves like with **M144**, but you cannot use cutter radius compensation outside of cycles.	Page 576
FUNCTION TCPM with **REFPNT TIP-CENTER**	HEIDENHAIN recommends using **FUNCTION TCPM** with **REFPNT TIP-CENTER**.	Page 393
	With **FUNCTION TCPM** and **REFPNT TIP-CENTER** selected, the tool location point is at the tool tip. The tool center of rotation is located at the tool center point.	Page 199
	If you activate **FUNCTION TCPM** with **REFPNT TIP-CENTER**, a tool-tip radius compensation is possible in positioning blocks with **RL/RR**.	
Cycle **800**	Use Cycle **800 ADJUST XZ SYSTEM** to define an inclination angle.	See the User's Manual for Machining Cycles

If you execute turning cycles with the functions stated above, then the angles of the tool relative to the contour will change. The control automatically takes these modifications into account and therefore also monitors the inclined machining operation.

Notes

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

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422

6.2.4 Simultaneous turning

Application

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

Related topics

- Simultaneous turning cycles (#158 / #4-03-2)

  Further information: User's Manual for Machining Cycles

- M function **M128** (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 576

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

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

Requirements

- Machine with at least two rotary axes
- Software option Turning (#50 / #4-03-1)
- Adv. Function Set 2 (#9 / #4-01-1) software option

Description of function

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

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

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

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

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

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

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

- The tool must be measured at the center of the tool-tip radius

Further information: "Presets on the tool", Page 199

Example

An NC program with simultaneous turning includes the following components:

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

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

6.2.5 Turning operations with FreeTurn tools

Application

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

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

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

Related topics

- Inclined turning

  Further information: "Inclined turning", Page 162
- Simultaneous turning operation

  Further information: "Simultaneous turning", Page 164
- FreeTurn tools

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

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

Requirements

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

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

  The control inclines the cutting edge by means of inclining the tool spindle.
- Software option Turning (#50 / #4-03-1)
- Kinematics description

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

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

Description of function
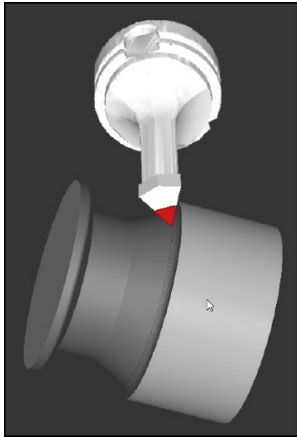
FreeTurn tool in simulation

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

Further information: User's Manual for Machining Cycles

FreeTurn tools

FreeTurn indexable insert for roughing
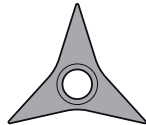
FreeTurn indexable insert for finishing
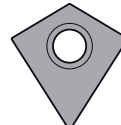
FreeTurn indexable insert for roughing and finishing

The control supports all variants of FreeTurn tools:

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

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

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

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

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

FreeTurn tool carrier

Tool carrier template for a FreeTurn tool

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

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

Notes

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

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

6.2.6   Unbalance compensation in turning operations

Application

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

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

Related topics

- Determining the unbalance of the current fixture

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

- Cycle **892 CHECK UNBALANCE**

  Further information: User's Manual for Machining Cycles

- Cycle **239 ASCERTAIN THE LOAD** (option 143)

  Further information: User's Manual for Machining Cycles

Description of function

> ⚙ Refer to your machine manual.
> Unbalance functions are not required and available on all machine tool types.
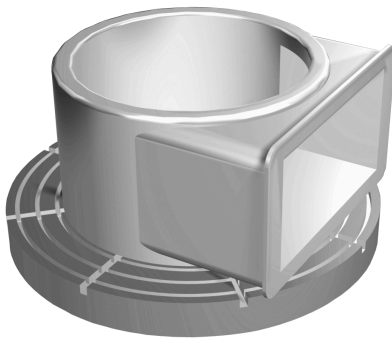
The centrifugal force that occurs basically depends on the rotational speed, the mass and the unbalance of the workpiece. A body with an uneven mass distribution that is put into rotary motion produces an unbalance. If the mass object is rotating, this creates outward-acting centrifugal forces. If the rotating mass is evenly distributed, the centrifugal forces cancel each other out. You compensate for the arising centrifugal forces by attaching compensation weights.

The control provides the **MEASURE UNBALANCE** cycle for this purpose. The cycle determines the existing unbalance and calculates the mass and position of the required balancing mass.

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

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

Further information: User's Manual for Machining Cycles

Unbalance monitor

The Unbalance Monitor function monitors the unbalance of a workpiece in turning mode. If a maximum unbalance limit specified by the machine manufacturer is exceeded, the control issues an error message and initiates an emergency stop.

The control automatically activates the Unbalance Monitor function when you switch to turning mode. The unbalance monitor is active until you switch back to milling mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 154

Notes

> ⚠ **WARNING**
>
> **Caution: Danger to the operator and machine!**
>
> Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!
>
> ▶ Clamp the workpiece in the spindle center
> ▶ Clamp workpiece securely
> ▶ Program low spindle speeds (increase as required)
> ▶ Limit the spindle speed (increase as required)
> ▶ Eliminate unbalance (calibrate)

■ The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.

■ The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps.

6.3 Grinding operations (#156 / #4-04-1)

6.3.1 Fundamentals

Special types of milling machines allow performing both milling and grinding operations. A workpiece can thus be machined completely on one machine, even if complex milling and grinding operations are required.

Requirements

- Grinding (#156 / #4-04-1) software option
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Production processes

The term grinding encompasses many types of machining that differ in quite a few respects, e.g.:

- Jig grinding
  Jig grinding is the grinding of a 2D contour. The tool movement in the plane is optionally superimposed by a reciprocation movement along the active tool axis.
  Further information: "Jig grinding", Page 174
- Cylindrical grinding
  Cylindrical grinding refers to the grinding of workpieces that are rotationally symmetrical. During cylindrical grinding, the tool in the tool spindle rotates, as does the clamped workpiece on the rotary table.
  Further information: "Cylindrical grinding", Page 175
- Surface grinding
  Surface grinding refers to the grinding of flat surfaces.
  Surface grinding is currently not available on the TNC7.

If grinding is enabled on your milling machine, (#156 / #4-04-1), the dressing function is also available. This means that you can shape or resharpen the grinding wheel in the machine.

Further information: "Dressing", Page 178

Reciprocating stroke

For jig grinding, the movement of the tool in the plane can be superimposed by a stroke movement, the so-called reciprocating stroke. The superimposed stroke movement is applied in the active tool axis.

You define an upper and a lower stroke limit and can start and stop the reciprocating stroke and reset the corresponding values. The reciprocating stroke is active until you stop it. **M2** or **M30** will stop the reciprocating stroke automatically.

The control provides cycles for defining, starting, and stopping reciprocating strokes.

For cylindrical grinding, the control automatically determines the reciprocating stroke from the data in the definition and infeed cycles.

As long as the reciprocating stroke is active during program run. you can't switch to the **Manual** operating mode.

The control shows the reciprocating stroke in the **Simulation** workspace of the **Program Run** operating mode.

Tools for grinding

When managing grinding tools, other geometric descriptions than those for milling or drilling tools are required. The control provides a special tool table for grinding and dressing tools. In the **Form** workspace of the tool management, the control displays only the required parameters for the current tool type.

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

You can use compensation tables to change the values of grinding tools during program run.

Further information: "Tool compensation with compensation tables", Page 418

Grinding wheel edges and teeth of grinding tools

For dressing and cylindrical grinding you must select a grinding wheel edge.

For cylindrical grinding you choose which grinding wheel edge the control will use for positioning. For dressing you choose which grinding wheel edge the control will dress.

The grinding tool is viewed in the cutting plane with different grinding wheel edges. You select the grinding wheel edge with Cycle **1030 ACTIVATE WHEEL EDGE** or in some of the cylindrical grinding cycles.

Further information: User's Manual for Machining Cycles

Depending on the type of grinding tool and the machining mode, you can select the following grinding wheel edges:

Grinding tool type	Grinding	Dressing
▪ **Cylindrical grinding pin** ▪ **Straight wheel** ▪ **Facing wheel**		
▪ **Conical grinding pin** ▪ **Angular wheel**		
Cup wheel		

When using inclined machining, you choose a tooth to be machined in addition to the grinding wheel edge. The straight line between two grinding wheel edges forms one tooth.

In the cylindrical grinding cycles the control displays the possible grinding wheel edges and teeth in a selection window.

Further information: User's Manual for Machining Cycles

The grinding wheel edges are either at the intersections of the adjoining teeth or at the intersection of the tooth with the tool axis. If the grinding wheel edge has a radius, the control still positions to the intersection of the adjoining teeth.

> **i** When changing the tool with a **TOOL CALL**, by default the control activates grinding wheel edge **9**.

6.3.2 Jig grinding

Application

On a milling machine, jig grinding will mainly be used for finishing a pre-machined contour with a grinding tool. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin or a grinding wheel. Jig grinding produces more precise results and a better surface definition than milling.

Related topics

- Cycles for grinding
  Further information: User's Manual for Machining Cycles
- Parameters for grinding tools
  Further information: User's Manual for Setup and Program Run
- Dressing of grinding tools
  Further information: "Dressing", Page 178

Requirements

- Grinding (#156 / #4-04-1) software option
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Description of function

Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Grinding is also possible with a tilted working plane. The tool reciprocates along the active tool axis in the current working plane coordinate system (**WPL-CS**).

Notes

- The control does not support block scans while the reciprocating stroke is active.
  Further information: User's Manual for Setup and Program Run
- The reciprocating stroke continues to be in effect during a programmed **STOP** or **M0** as well as in **Single Block** mode, even after the end of an NC block.
- If no cycle has been programmed and a contour is being ground whose smallest inside radius is smaller than the tool radius, the control will display an error message.
- If you machine with SL cycles, only those areas will be ground that are suitable for the given tool radius. In this case, the resulting contour will not be completely finished and may need to be reworked.

Program structure for jig grinding

Here you see a possible program structure for jig grinding.

	BLK FORM...	
	TOOL CALL...	
Dressing of grinding tools (if applicable)	**...**	; Macro or **FUNCTION DRESS**
Defining the reciprocating stroke	**CYCL DEF 1000 DEFINE RECIP. STROKE**	
If necessary, explicitly starting the reciprocating stroke	**CYCL DEF 1001 START RECIP. STROKE**	
Machining a contour	**CYCL DEF 14 CONTOUR**	
	CYCL DEF 1025 GRINDING CONTOUR	
	CYCL CALL...	
Stopping the reciprocating stroke	**CYCL DEF 1002 STOP RECIP. STROKE**	
...		

> ℹ️ You can use specific machining cycles (e.g., cycles for grinding, for machining pockets or studs, or SL cycles) to define the contour.

More detailed information

- Dressing

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

- Cycles for dressing

  Further information: User's Manual for Machining Cycles

- Cycles for Grinding

  Further information: User's Manual for Machining Cycles

6.3.3 Cylindrical grinding

Application

Cylindrical grinding on a milling machine enables you to perform complete machining tasks on a machine without rechucking. Cylindrical grinding helps you to attain higher accuracies and better surface definitions than with turning.

Related topics

- Cycles for grinding
  Further information: User's Manual for Machining Cycles
- Parameters for grinding tools
  Further information: User's Manual for Setup and Program Run
- Dressing of grinding tools
  Further information: "Dressing", Page 178
- Turning (#50 / #4-03-1)
  Further information: "Turning operations (#50 / #4-03-1)", Page 157

Requirements

- Grinding (#156 / #4-04-1) software option
- Machine with at least two rotary axes, one of them as a rotary table axis
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Description of function

Machining is performed in the cylindrical grinding mode **FUNCTION MODE GRIND**.

Further information: "Switching the operating mode with FUNCTION MODE", Page 154

Depending on the machining direction and the actual task, cylindrical grinding can include the following operations, among others:

- Long-stroke cylindrical grinding
- Short-stroke cylindrical grinding
- Plunge grinding
- Shoulder grinding
- Multiple recessing

The control offers cycles for these operations. These cycles can also be used with inclined rotary axes.

For cylindrical grinding operations you always program a definition cycle, at least one infeed cycle, and a conclusion cycle.

Further information: "Program structure for cylindrical grinding", Page 177

Coordinate plane for cylindrical grinding

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

This means that you always program with **ZX** as the machining plane. The kinematics of the respective machine determine which axes the machine uses for the actual movements. NC programs with cylindrical grinding cycles are largely independent of the machine kinematics.

Notes

- Refer to your machine manual.
  If your machine tool is equipped with an acoustic-emission sensor, the control can evaluate the signal in the infeed cycles. The control can thus prevent reciprocating strokes from taking place in air.
  Further information: User's Manual for Machining Cycles
- Dress the grinding tool either before or after a cylindrical grinding operation.
  Further information: "Dressing", Page 178

Program structure for cylindrical grinding

Here you see a possible program structure for cylindrical grinding.

	BLK FORM...	
	TOOL CALL...	
Dressing of grinding tools (if applicable)	**...**	; Macro or **FUNCTION DRESS**
Definition cycle	**CYCL DEF 1041 LONG STROKE DEF.**	
	CYCL CALL...	
Infeed cycle	**CYCL DEF 1051 STEP. CYLIND. GRIND**	; At least one infeed cycle is necessary (e.g., roughing)
	CYCL CALL...	
	CYCL DEF 1051 STEP. CYLIND. GRIND	; Optional additional infeed cycle (e.g., finishing)
	CYCL CALL...	
	CYCL DEF 1051 STEP. CYLIND. GRIND	; Optional additional infeed cycle (e.g., fine finishing)
	CYCL CALL...	
Conclusion cycle	**CYCL DEF 1040 END CYLIND. GRINDING**	
	CYCL CALL...	
...		

More detailed information

- Dressing

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

  Further information: User's Manual for Machining Cycles

6.3.4   Inclined cylindrical grinding

Application

It may sometimes be necessary for you to bring the rotary axes into a specific position to machine a specific process. This can be necessary, for example, when you can machine contour elements at only a certain inclination of the grinding wheel.

Related topics

- Inclined machining

  Further information: "Inclined machining", Page 542

Requirements

- Machine with at least two rotary axes, one of them as a rotary table axis
- Available kinematics description for jig grinding

  The machine manufacturer creates the kinematics description.

- Grinding (#156 / #4-04-1) software option

Description of function
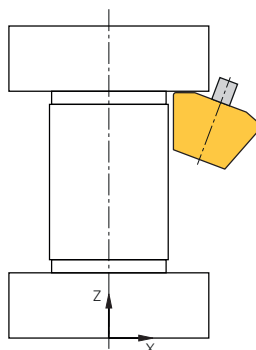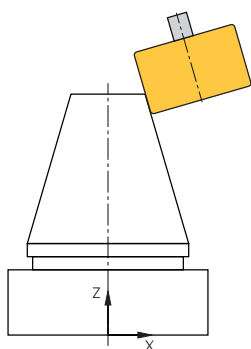
Inclined cylindrical grinding is possible with Cycle **1041 LONG STROKE DEF.**.

Further information: User's Manual for Machining Cycles

As an alternative you can use Cycle **800 ADJUST XZ SYSTEM** to define an inclination for cylindrical grinding.

Further information: User's Manual for Machining Cycles

Notes

- You must select the desired grinding wheel edge before inclining the tool.

  Further information: "Grinding wheel edges and teeth of grinding tools", Page 173

  Further information: User's Manual for Machining Cycles

- The **FUNCTION CORRDATA TCS** tool compensation is always in effect in the tool coordinate system, even during inclined machining.

  Further information: "Tool compensation with compensation tables", Page 418

6.3.5   Dressing

Application

The term "dressing" refers to the sharpening or truing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, during dressing, the grinding tool is the workpiece.

Related topics

- Activating dressing mode with **FUNCTION DRESS**

    Further information: "Activating dressing mode with FUNCTION DRESS", Page 181

- Cycles for dressing

    Further information: User's Manual for Machining Cycles

- Parameters for dressing tools

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

- Jig grinding

    Further information: "Jig grinding", Page 174

- Cylindrical grinding

    Further information: "Cylindrical grinding", Page 175

Requirements

- Grinding (#156 / #4-04-1) software option
- Available kinematics description for jig grinding

    The machine manufacturer creates the kinematics description.

Description of function

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge using Cycle **1030 ACTIVATE WHEEL EDGE**.

During dressing, the axes are arranged such that the X coordinates describe positions on the radius of the grinding wheel, and the Z coordinates describe the positions along the axis of the grinding wheel. The dressing programs are thus not contingent on the machine type.

The machine manufacturer defines which machine axes will perform the programmed movements.

The dressing operation removes material from the grinding wheel and may cause wear of the dressing tool. The material removal and wear lead to changed parameters that need to be compensated for after dressing.

The **COR_TYPE** parameter provides the following compensation options:

- **Grinding wheel with compensation, COR_TYPE_GRINDTOOL**

    Compensation method with material removal at grinding tool

    Further information: "Stock removal on the grinding tool", Page 180

- **Dressing tool with wear, COR_TYPE_DRESSTOOL**

    Compensation method with material removal at dressing tool

    Further information: "Stock removal on the grinding tool", Page 180

Use the Cycles **1032 GRINDING WHL LENGTH COMPENSATION** and **1033 GRINDING WHL RADIUS COMPENSATION** to compensate for the grinding wheel or the dresser, regardless of the compensation method.

Dressing a grinding wheel edge shape

You can define a grinding wheel edge shape for the following types of grinding tools:

■ **Cylindrical grinding pin**

■ **Straight wheel**

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

The control provides the following possibilities for dressing the defined grinding wheel edge shape:

■ Only the front face or shaft face with Cycle **1011 DRESSING SIDE A/I**

■ Front face or shaft face and diameter with Cycle **1012 DRESSING D AND A/I**

Further information: User's Manual for Machining Cycles

Simplified dressing with a macro

| **i** | Refer to your machine manual. |

The machine manufacturer can integrate a macro in the dressing cycles. The macro handles, for example, selection of the dressing mode **FUNCTION DRESS** and the grinding wheel edge, and also the **TOOL CALL** for the dressing tool. In this case, the machine manufacturer determines the dressing sequence.

Compensation methods

Stock removal on the grinding tool

During dressing, a dressing tool is usually used that is harder than the grinding tool. Due to the difference in hardness, the stock removal during dressing mainly takes place at the grinding tool. The programmed dressing amount is actually removed at the grinding tool, since the dressing tool does not noticeably wear. In this case the compensation method **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** is used in the **COR_TYPE** parameter of the grinding tool.

With this compensation method, the parameters of the dressing tool remain constant. The control compensates only for the grinding tool:

■ Programmed dressing amount in the basic data of the grinding tool (e.g., **R-OVR**)

■ If applicable, measured deviation between nominal and actual dimension in the compensation data of the grinding tool (e.g., **dR-OVR**)

Stock removal on dressing tool

In contrast to the standard situation, stock removal does not take place only on the grinding tool in certain grinding and dressing combinations. In this case the dressing tool wears noticeably (for example, with very hard grinding tools in combination with softer dressing tools). To compensate for this noticeable wear on the dressing tool, the control offers the compensation method **Dressing tool with wear, COR_TYPE_DRESSTOOL** in the **COR_TYPE** parameter of the dressing tool.

With this compensation method the parameters of the dressing tool change significantly. The control compensates for both the grinding tool and the dressing tool:

- Dressing amount in the basic data of the grinding tool (e.g., **R-OVR**)
- Measured wear in the compensation data of the dressing tool (e.g., **DXL**)

If you use the compensation method **Dressing tool with wear, COR_TYPE_DRESSTOOL**, the control stores the tool number of the dressing tool used in the **T_DRESS** parameter of the grinding tool after dressing. During future dressing processes, the control monitors whether the defined dressing tool is used. If you use a different dressing tool, the control interrupts the dressing with an error message.

You must recalibrate the grinding tool after each dressing process so that the control can determine and compensate for the wear.

Notes

- For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.
- When using the **Dressing tool with wear, COR_TYPE_DRESSTOOL** correction method, inclined dressing tools must not be used.
- When dressing a grinding wheel with relief cut, the angle of infeed **Q1023** must be equal to or greater than the relief cut. If you program a smaller angle of infeed, the grinding wheel will lose dimensional accuracy.
- Dress the grinding tool either before or after a cylindrical grinding operation.

6.3.6 Activating dressing mode with FUNCTION DRESS

Application

With **FUNCTION DRESS** you activate a dressing kinematic model for dressing a grinding tool. The grinding tool is then the workpiece and the axes may move in the opposite direction.

Your machine manufacturer might provide a simplified dressing procedure.

Further information: "Simplified dressing with a macro", Page 180

Related topics

- Cycles for dressing

  Further information: User's Manual for Machining Cycles
- Fundamentals of dressing

  Further information: "Dressing", Page 178

Requirements

- Grinding (#156 / #4-04-1) software option
- Available kinematics description for dressing
  The machine manufacturer creates the kinematics description.
- Grinding tool is inserted

Description of function

> ### *NOTICE*
>
> **Danger of collision!**
>
> When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!
>
> ▶ Activate the **FUNCTION DRESS** dressing mode only in the **Program Run** operating mode or in **Single Block** mode
> ▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
> ▶ Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine manufacturer
> ▶ In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
> ▶ If necessary, program a kinematic switch-over

For the control to switch to the kinematic model for dressing, you must program the dressing process between the functions **FUNCTION DRESS BEGIN** and **FUNCTION DRESS END**.

If dressing mode is active, the control shows a corresponding symbol in the **Positions** workspace.

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

You can switch back to normal operation with the function **FUNCTION DRESS END**.

In the event of an NC program abort or a power interruption, the control automatically activates normal operation and the kinematic model that was active prior to dressing mode.

Input

11 FUNCTION DRESS BEGIN "Dress"	; Activate dressing mode with the **Dress** kinematics

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Dressing DRESS

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DRESS	Syntax initiator for dressing mode
BEGIN or **END**	Activate or deactivate dressing mode
Name or **Parameter**	Name of the selected kinematic model
	Text or string parameter
	Optional syntax element
	Selection by means of a selection window
	Only if **BEGIN** has been selected

Notes

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! There is a danger of collision!

▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool

▶ Make sure there is no risk of collision

▶ Slowly prove-out the NC program

NOTICE

Danger of collision!

With an active kinematic model, the machine movements may be in the opposite direction. There is a risk of collision when moving the axes!

▶ In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes

▶ If necessary, program a kinematic switch-over

■ During dressing, the cutting edge of the dresser must be at the same height as the grinding wheel. The programmed Y coordinate must be 0.

■ With the switch to dressing mode, the grinding tool remains in the spindle and retains its current rotational speed.

■ The control does not support a block scan during the dressing process. If, during a block scan, you select the first NC block after the dressing operation, then the control moves to the most recently approached position in the dressing operation.

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

■ If the "tilt working plane" function or **TCPM** function is active, then you cannot switch to dressing mode.

■ The control resets the manual tilting functions (#8 / #1-01-1) and the function **FUNCTION TCPM** (#9 / #4-01-1) when it activates dressing mode.

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

   Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

■ In dressing mode you can use **TRANS DATUM** to change the workpiece datum. No other NC functions or coordinate conversion cycles are permitted in dressing mode. The control displays an error message.

   Further information: "Datum shift with TRANS DATUM", Page 334

■ The **M140** function is not allowed in dressing mode. The control displays an error message.

■ The control does not graphically depict the dressing operation. The times determined by the simulation do not reflect the actual machining times. One reason for this is the necessary switching of the kinematic model.

7

Workpiece Blank

7.1 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 761

■ Workpiece blank for turning **FUNCTION TURNDATA BLANK** (#50 / #4-03-1)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422

Description of function

You define the blank relative to the workpiece preset.

Further information: "Presets in the machine", Page 126
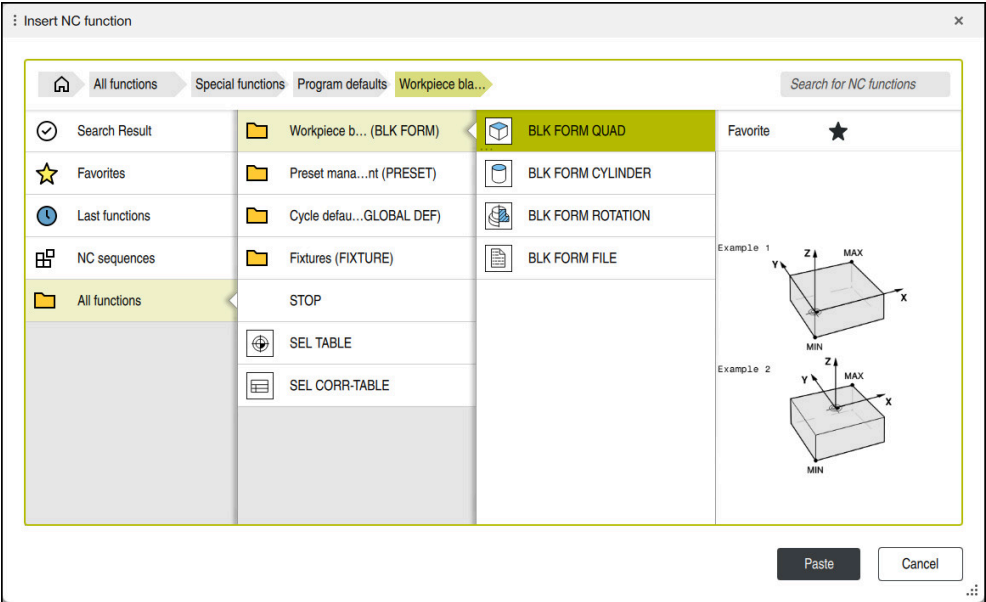
The **Insert NC function** window for workpiece blank definition

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 105

The control offers the following workpiece blank definitions:

Icon	Meaning	Further information
	BLK FORM QUAD Cuboid workpiece blank	Page 189
	BLK FORM CYLINDER Cylindrical workpiece blank	Page 190
	BLK FORM ROTATION Rotationally symmetric blank with a definable contour	Page 191
	BLK FORM FILE STL file as workpiece blank and finished part	Page 193

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 to select 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 string parameter
  ■ Define the number of the subprogram in a numerical 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.

■ The control does not use the **BLK FORM** function to generate the traverse paths for the turning cycles (#50 / #4-03-1). In this case, define **FUNCTION TURNDATA BLANK**.

  Further information: "Blank form update in turning with FUNCTION TURNDATA BLANK (#50 / #4-03-1)", Page 194

■ If you have closed the **Insert NC function** window or want to add a workpiece blank definition after having created an NC program, a workpiece blank can be defined at any time in 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 481

■ 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 774

7.1.1 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
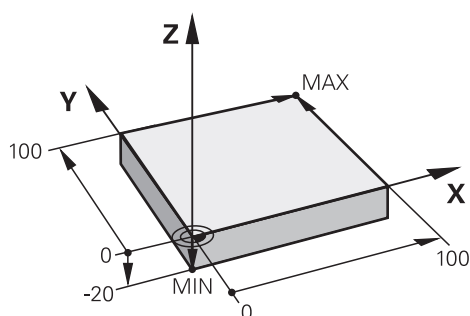
Cuboid workpiece blank with MIN point and MAX point

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 126

If you use a cuboid workpiece blank for turning (#50 / #4-03-1), keep the following in mind:

Even if the turning operation takes place in a two-dimensional plane (Z and X coordinates), you have to program the Y values for a rectangular blank in the definition of the workpiece blank.

Further information: "Fundamentals", Page 157

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

To navigate to this function:

Insert NC function ▶ Special functions ▶ Program defaults ▶ Workpiece blank definition (BLK FORM) ▶ BLK FORM QUAD

The NC function includes the following syntax elements:

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

7.1.2 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
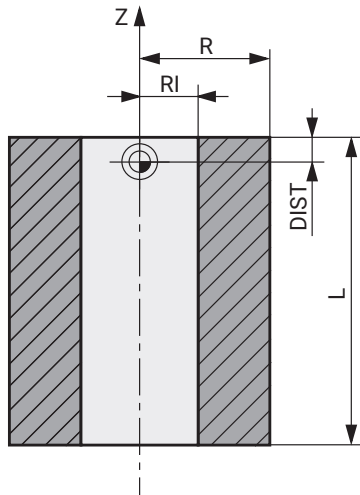
Cylindrical blank

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 ▶ Workpiece blank definition (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

7.1.3 Rotationally symmetric workpiece blank with BLK FORM ROTATION

Application

With the **BLK FORM ROTATION** function you define a rotationally symmetrical workpiece blank in order to, for example, correctly simulate machining on a fully turned shaft.

Description of function

For a rotationally symmetrical workpiece blank you define one axis as the rotational axis. The rotational axis defines the coordinate plane for the workpiece blank's contour description (e.g., the Z/X plane)
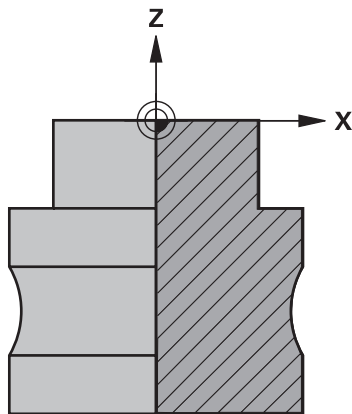
Workpiece blank contour with rotational axis **Z** and secondary axis **X**

In the workpiece blank definition you refer to the contour description.

You program the contour in a subprogram or in a separate NC program.

You program the half-section of the workpiece blank as a contour. The contour of the half-section rotates around the rotational axis.

The following conditions apply to the contour description:

- Use only coordinates in the plane

  If **Z** is the rotational axis, you program the contour of the workpiece blank in the Z/X plane. **Z** is then the main axis and **X** is the secondary axis.

- For the starting point, always program both coordinates in the plane
- Always program a closed contour
- Program only positive values in the secondary axis

The workpiece preset is on the rotational axis. You define the coordinates of the blank contour relative to the workpiece preset.

The workpiece preset does not need to be on the surface of the workpiece; it can also be within the workpiece. This allows you to define an oversize, for example.

> ℹ️ The positive direction of rotation for circular contour elements always goes from the main axis to the secondary axis; for example, from **Z** to **X**.
>
>
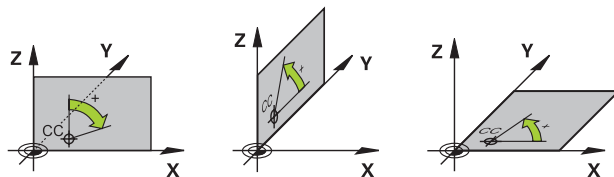
Input

1 BLK FORM ROTATION Z DIM_D LBL "BLANK"	; Rotationally symmetrical workpiece blank with rotational axis **Z**
* - ...	
11 M30	
12 LBL "BLANK"	; Subprogram start
13 L X+0 Z+0	; Contour starting point
14 L X+35	; Coordinates in positive direction of secondary axis
15 L Z-15	
16 L X+50	
17 L Z-25	
18 CR X+50 Z-40 R+15 DR-	
19 L Z-50	
20 L X+0	
21 L Z+0	; Contour end point identical to contour starting point
22 LBL 0	; End of subprogram
* - ...	

To navigate to this function:

Insert NC function ▶ Special functions ▶ Program defaults ▶ Workpiece blank definition (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, **X** or **Y**	Rotary axis The rotational axis defines the coordinate plane for the contour description.
DIM_R or **DIM_D**	Interpret values in the secondary axis 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 CAD Import software option (#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

7.1.4 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 ▶ Workpiece blank definition (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 774

- 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 780

- 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.

7.2 Blank form update in turning with FUNCTION TURNDATA BLANK (#50 / #4-03-1)

Application

Using the blank form update feature, the control detects the already machined areas and adapts all approach and departure paths to the specific, current machining situation. Thus, air cuts are avoided and the machining time is significantly reduced.

You define the workpiece blank for blank form update in a subprogram or separate NC program.
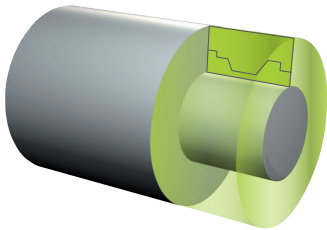
Related topics

- Subprograms

  Further information: "Subprograms and program section repeats with the label LBL", Page 282
- Turning mode: **FUNCTION MODE TURN**

  Further information: "Fundamentals", Page 157
- Defining a workpiece blank with **BLK FORM** for simulation

  Further information: "Defining a workpiece blank with BLK FORM", Page 186

Requirements

- Software option Turning (#50 / #4-03-1)
- **FUNCTION MODE TURN** must be active

  Blank form update is only possible with cycle machining in turning mode.
- Closed blank contour for blank form updating

  The starting and end positions must be identical. The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.

Description of function
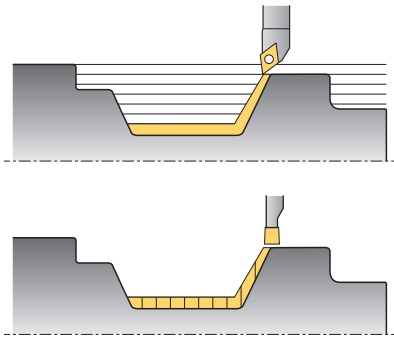
With **TURNDATA BLANK** you call a contour description used by the control as an updated workpiece blank.

You can define the workpiece blank in a subprogram within the NC program or as a separate NC program.

Blank form update is only active in conjunction with roughing cycles. In finishing cycles the control always machines the entire contour, for example so that the contour does not have any offset.

If the contour to be machined is larger than the workpiece blank, the control will display an error message.

Further information: User's Manual for Machining Cycles

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 string parameter
- Define the number of the subprogram in a numerical parameter

Use **FUNCTION TURNDATA BLANK OFF** to deactivate blank form update.

Input

1 FUNCTION TURNDATA BLANK LBL "BLANK"	; Blank form update with a workpiece blank from the subprogram "BLANK"
* - ...	
11 LBL "BLANK"	; Subprogram start
12 L X+0 Z+0	; Beginning of contour
13 L X+50	; Coordinates in positive direction of main axis
14 L Z+50	
15 L X+30	
16 L Z+70	
17 L X+0	
18 L Z+0	; End of contour
19 LBL 0	; End of subprogram

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Turning functions** ▶ **Basic functions** ▶ **FUNCTION TURNDATA BLANK**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA BLANK	Syntax initiator for blank form update in turning mode
OFF, **File**, **QS**, or **LBL**	Deactivate blank form update, blank contour as separate NC program, or call as subprogram
Number, **Name** or **Parameter**	Number or name of the separate NC programor subprogram Number, text, or variable Selection by means of a selection window When **File**, **QS**, or **LBL** is selected

8

Tools

8.1 Fundamentals

To use all of the control's functions, you must define the tools for the control using real data (e.g., the radius). This increase process reliability.

To add a tool to the machine and then be able to use it, follow the sequence below:

- Clamp the tool in an appropriate 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 199
- Further parameters are needed to completely define the tool. One place to find these parameters is the manufacturer's tool catalog.

  Further information: User's Manual for Setup and Program Run
- Save all collected parameters 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: "Using TOOL CALL to call a tool", Page 205
- 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 using TOOL DEF", Page 213
- 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 412

8.2 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 126

8.2.1 Tool carrier reference point
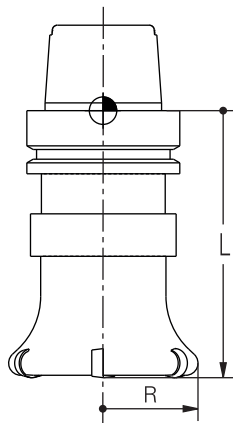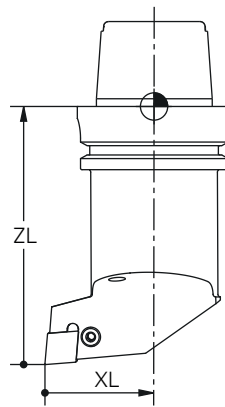
Milling cutter   Turning tool   Grinding tool
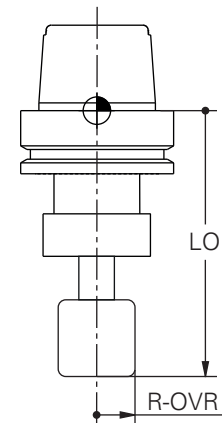
The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

8.2.2   Tool tip TIP
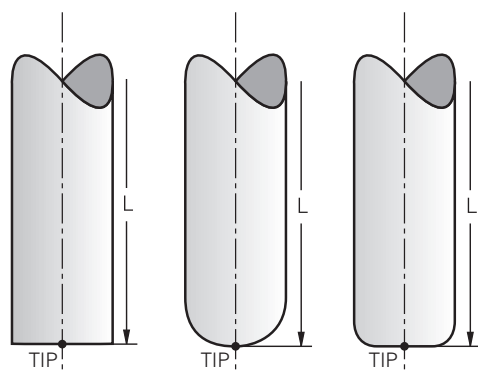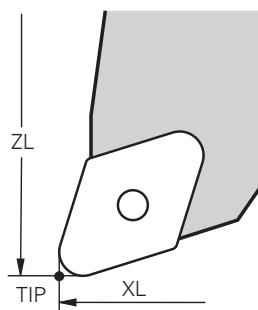
Milling tools   Lathe tools
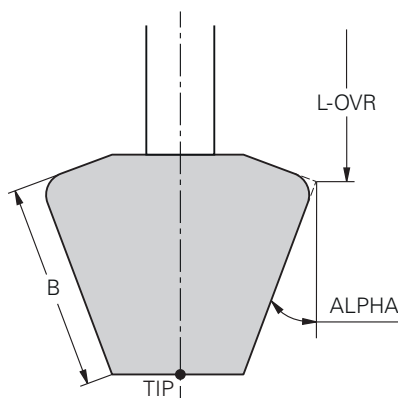
Grinding tools

Further information: "Tool coordinate system T-CS", Page 323

You define the position of the tool tip with the basic and delta values of the tool relative to the tool-carrier reference point.

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

In case of milling cutters, the tool tip is at the center of the tool diameter and at the longest point of the tool on the tool axis.

For turning tools (#50 / #4-03-1), the control uses the theoretical tool tip, i.e. the longest measured values for **ZL**, **XL**, and **YL**.

In case of grinding tools (#156 / #4-04-1), the tool tip is at the center of the tool diameter and at the longest point of the tool on the tool axis.

For the following grinding tools, the control calculates the longest point of the tool from several parameters:

- **Angular wheel**

  L-OVR, **ALPHA** and **B**
- **Straight wheel** and **Facing wheel**

  L-OVR and **B**

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 202

8.2.3 Tool center point (TCP, tool center point)
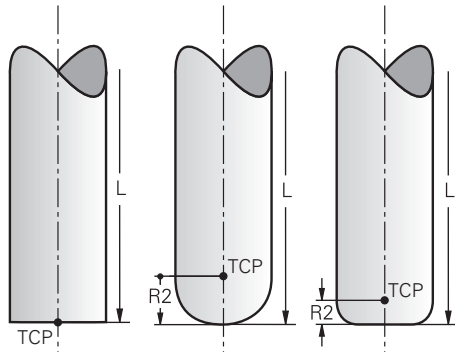
Milling tools
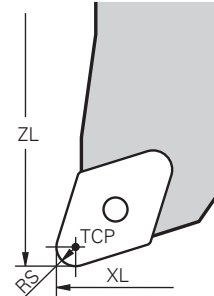
Lathe tools
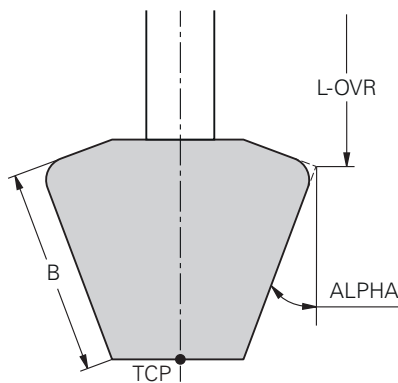
Grinding tools
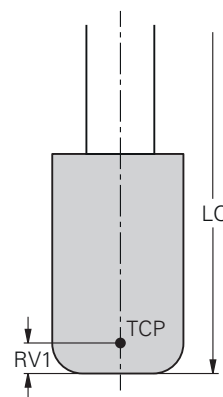
The tool center point TCP is the center of the tool diameter. If a tool radius 2 **R2** is defined, the tool center point is offset from the tool tip by this value.

For turning tools (#50 / #4-03-1), the tool center point lies at the center of the tool-tip radius **RS**.

If the radius of a grinding tool (#156 / #4-04-1) is defined at the lower tool edge **RV1**, 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 202

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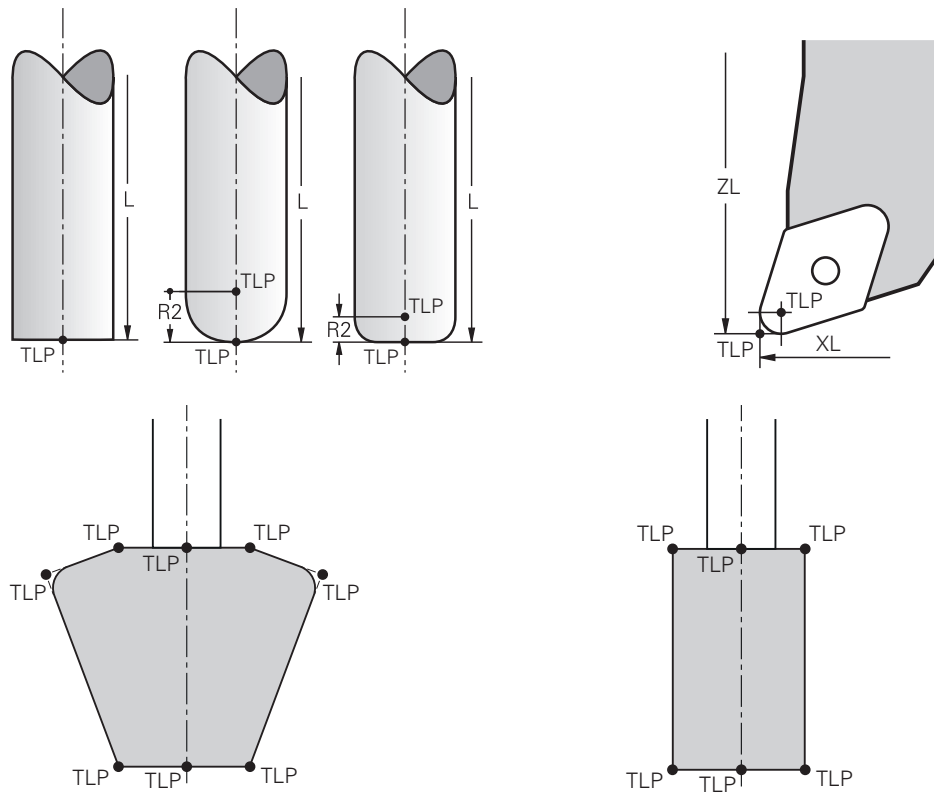
The control positions the tool on the tool location point TLP. By default, the tool location point is at the tool tip.

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 393

For cylindrical grinding (#156 / #4-04-1) you select a grinding wheel edge. The control sets the tool location point on the selected grinding wheel edge.

Further information: "Grinding wheel edges and teeth of grinding tools", Page 173

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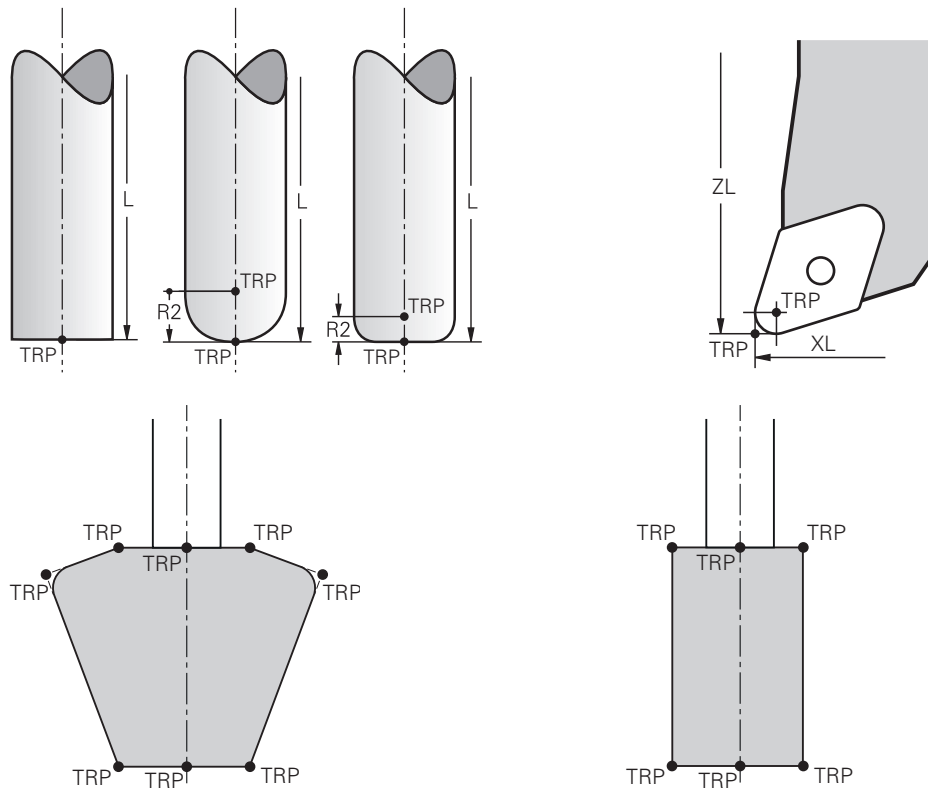
When applying the tilting function with **MOVE** (#8 / #1-01-1), the control tilts around the tool rotation point TRP. By default, the tool center of rotation is at the tool tip.

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 381

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 393

For cylindrical grinding (#156 / #4-04-1) you select a grinding wheel edge. The control sets the tool rotation point on the selected grinding wheel edge.

Further information: "Grinding wheel edges and teeth of grinding tools", Page 173

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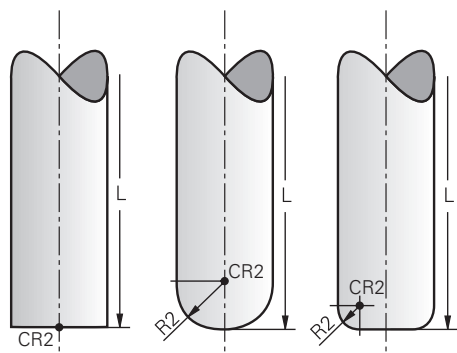
The control uses the tool radius 2 center in conjunction with 3D tool compensation (#9 / #4-01-1). In the case of straight lines **LN**, the surface-normal vector points to that point and defines the direction of the 3D tool compensation.

Further information: "3D tool compensation (#9 / #4-01-1)", Page 424

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 202

8.3 Tool call

8.3.1 Using TOOL CALL to call a tool

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 591

- 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

> **i** 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 555

Icons

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

Icon	Meaning
凵⃞	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 756

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 **Parameter**	Number or name of the tool Number, text, or variable ℹ️ 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 207
.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 207
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 210
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 211
DL	Delta value of tool length Optional syntax element **Further information:** "Tool compensation for tool length and tool radius", Page 408

Syntax element	Meaning
DR	Delta value of the tool radius
	Optional syntax element
	Further information: "Tool compensation for tool length and tool radius", Page 408
DR2	Delta value of the tool radius 2
	Optional syntax element
	Further information: "Tool compensation for tool length and tool radius", Page 408

Technology-dependent differences when calling tools

Milling cutter tool call

The following tool data of a milling cutter can be defined:

- Number or name of the 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 turning tool (#50 / #4-03-1)

The following parameters of a turning tool can be defined:

- Number or name of the tool
- Step index of the tool
- Feed rate

Calling a turning tool requires the number or the name of the tool.

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

Tool call for a grinding tool (#156 / #4-04-1)

The following parameters of a grinding tool can be defined:

- Number or name of the tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate

Calling a grinding tool requires the number or the name of the tool and the tool axis.

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

Tool call for a dressing tool (#156 / #4-04-1)

The following parameters of a dressing tool can be defined:

- Number or name of the tool
- Step index of the tool
- Feed rate

Calling a dressing tool requires the number or the name of the tool!

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

A dressing tool can be called only in dressing mode!

Further information: "Activating dressing mode with FUNCTION DRESS", Page 181

The dressing tool will not be mounted to the spindle. You need to mount the dressing tool manually to a pocket defined by the machine manufacturer. Additionally, you must define the tool in the pocket table.

Tool call for a workpiece touch probe

The following parameters of a workpiece touch probe can be defined:

- Number or name of the 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 parameters

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

In the cases below, the control updates the parameters of only 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 591

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 using TOOL DEF", Page 213

- 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 408

8.3.2 Cutting data

Application

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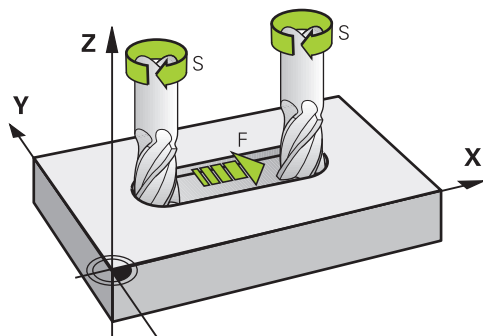
Description of function

Spindle speed S

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

- Tool call with **TOOL CALL**

  Further information: "Using TOOL CALL to call a tool", Page 205
- **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.

Further information: "Technology values for turning operations", Page 160

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 91

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: "Using TOOL CALL to call a tool", Page 205
- Positioning block

  Further information: "Path functions", Page 215
- **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.

  > **i** 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 per revolution is used mainly for turning (#50 / #4-03-1).

  Further information: "Feed rate", Page 161

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

Further information: "F AUTO", Page 211

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 91

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.

8.3.3 Tool pre-selection using 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 **Parameter**	Tool definition Number, text, or variable Selection by means of a selection window ℹ Only the tool definition as a number is unique because the tool names of several tools may be identical!
.1	Step index of the tool Optional syntax element **Further information:** User's Manual for Setup and Program Run

This function can be used for all technologies except for dressing tools (#156 / #4-04-1).

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

9

Path functions

9.1 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.

9.1.1 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.
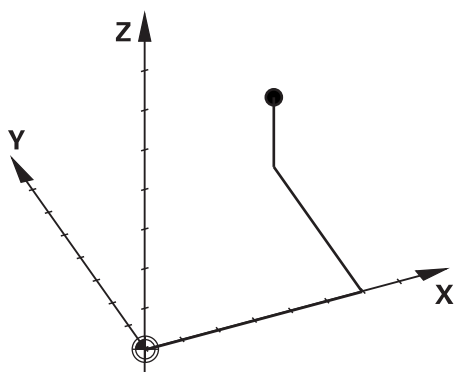
With Cartesian coordinates you can uniquely specify a point in space by defining the three axis values.

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.

9.1.2 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.
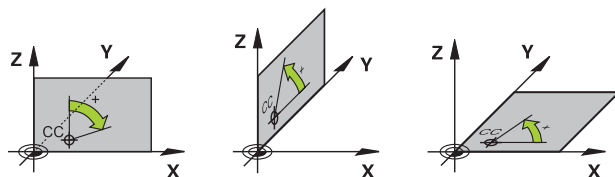
Description of function

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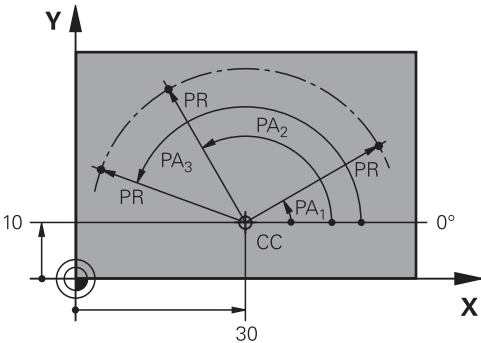
You define the pole **CC** with Cartesian coordinates in two axes. These axes specify the plane and the angle reference axis.

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.

9.1.3   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.
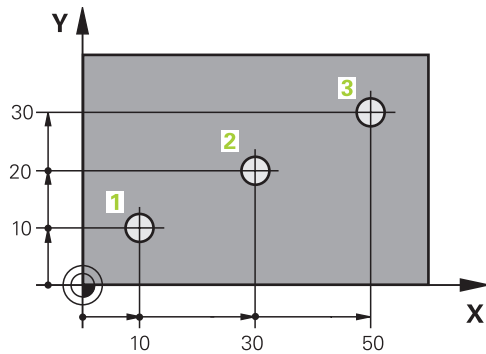
11 L X+10 Y+10 RL F200 M3	; Position at point 1
12 L X+30 Y+20	; Position at point 2
13 L X+50 Y+30	; Position at point 3
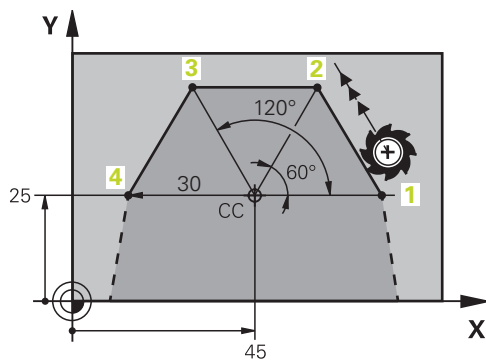
11 CC X+45 Y+25	; Define the pole with two axes using Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position at point 1
13 LP PA+60	; Position at point 2
14 LP PA+120	; Position at point 3
15 LP PA+180	; Position at point 4

9.1.4 Incremental entries

Application

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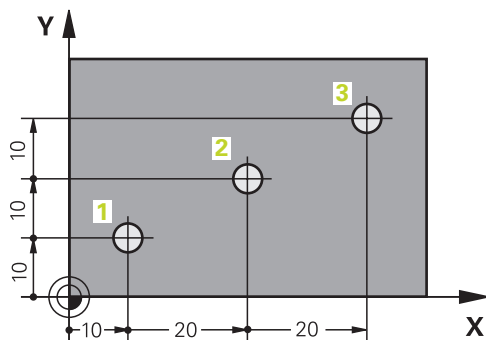
11 L X+10 Y+10 RL F200 M3	; Position to point 1 absolutely
12 L IX+20 IY+10	; Position to point 2 incrementally
13 L IX+20 IY+10	; Position to point 3 incrementally
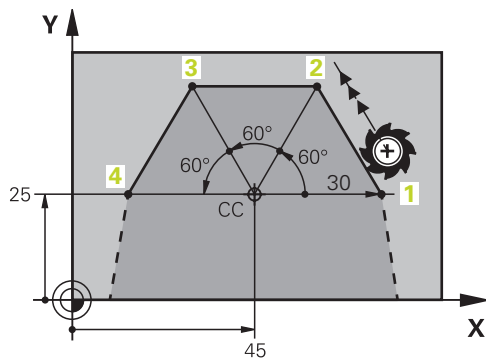
11 CC X+45 Y+25	; Define the pole absolutely in two axes with Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position to point 1 absolutely
13 LP IPA+60	; Position to point 2 incrementally
14 LP IPA+60	; Position to point 3 incrementally
15 LP IPA+60	; Position to point 4 incrementally

9.2 Fundamentals of path functions

Application

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

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

Description of function

Inserting a path function

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

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

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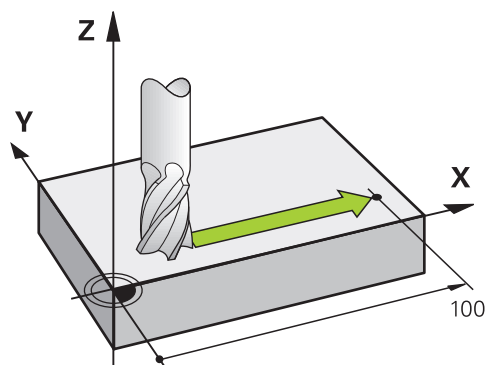
If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Example

```
L  X+100
```

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

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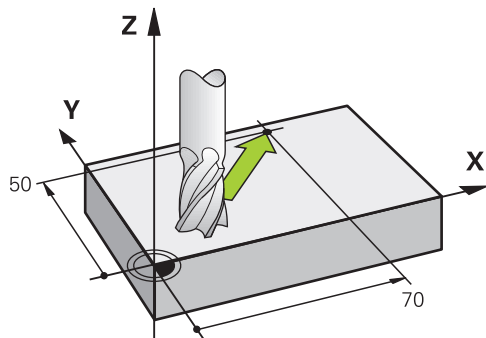
If the NC block contains two coordinates, the control moves the tool in the programmed plane.

Example

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

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

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

Further information: "Designation of the axes of milling machines", Page 124

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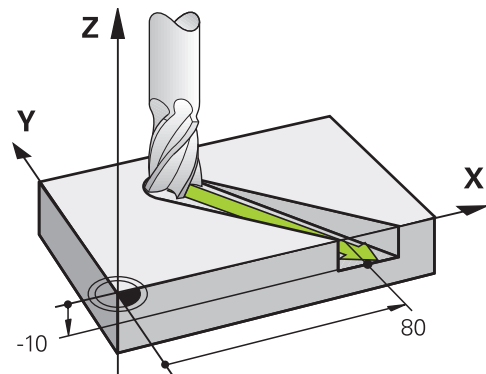
If the NC block contains three coordinate entries, the control moves the tool spatially to the programmed position.

Example

```
L X+80  Y+0 Z-10
```

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

Example

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

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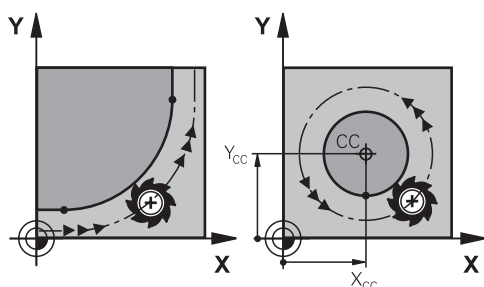
Use the path functions for circular arcs to program circular motions in the working plane.

The control moves the tool in two axes simultaneously on a circular path relative to the workpiece. You can program circular paths with a circle center point **CC**.

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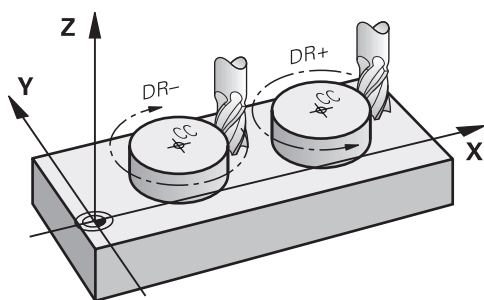
When a circular path has no tangential transition to another contour element, define the direction of rotation as follows:

■ Clockwise direction of rotation: **DR-**
■ Counterclockwise direction of rotation: **DR+**

Tool radius compensation

Tool radius compensation is defined in the NC block of the first contour element.

Do not activate tool radius compensation in an NC block for a circular path. Activate tool radius compensation in a preceding straight line.

Further information: "Tool radius compensation", Page 412

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

9.3 Path functions with Cartesian coordinates

9.3.1 Overview of path functions

Key	Function	Further information
L	Straight line **L** (line)	Page 224
CHF	Chamfer **CHF** (chamfer) Chamfer between two straight lines	Page 226
RND	Rounding **RND** (rounding of corner) Circular arc with tangential connection to the preceding and subsequent contour elements	Page 227
CC	Circle center point **CC** (circle center)	Page 229
C	Circular path **C** (circle) Circular path around a circle center **CC** to an end point	Page 230
CR	Circular path **CR** (circle by radius) Circular path with a specified radius	Page 232
CT	Circular path **CT** (circle tangential) Circular path with tangential connection to the preceding contour element	Page 235

9.3.2   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 243

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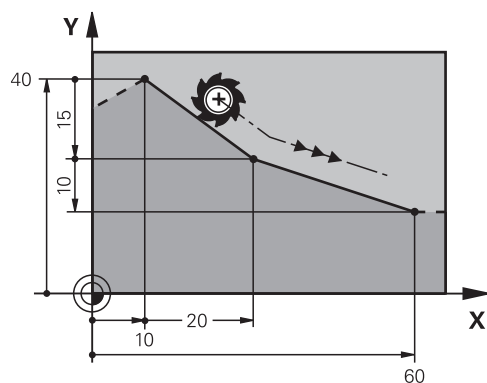
The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

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

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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
&X, **&Y**, **&Z**	End point of the straight line in a main axis deselected with **PARAXMODE**
	Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 520
	Number or numerical parameter
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 412
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	Optional syntax element

Straight line L with the values of the current position

To enter a straight line **L** with the values of the current position:

▶ Select the NC block after which you want to insert the straight line **L** block

⊕ ▶ Press the **actual position capture** key

 �> The control inserts a straight line **L** with the actual positions of all defined axes.

> ℹ️ ■ You use the **actPosAxes** machine parameter (no. 105415) to define the axes used by the **actual position capture** key to create a straight line **L**.
> ■ The values are equivalent to the **Actual pos. (ACT)** mode of the position display.

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 147

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

9.3.3 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
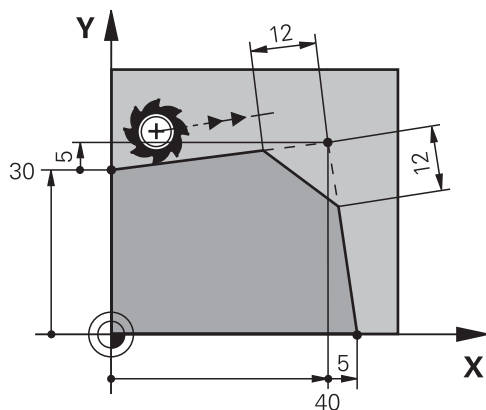
Cutting two straight lines creates contour corners. You can insert a chamfer at these contour corners. The angle of the corner is irrelevant; you simply define the length by which each straight line is shortened. The control does not traverse to the corner point.

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
	Number or numerical parameter
F, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	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

9.3.4 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
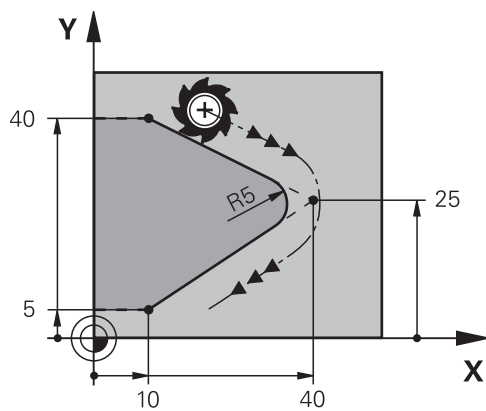
You program the rounding arc between two path functions. The circular arc connects tangentially to the previous and subsequent contour element. The control does not traverse to the intersection.

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
	Number or numerical parameter
F, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	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

9.3.5 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 241

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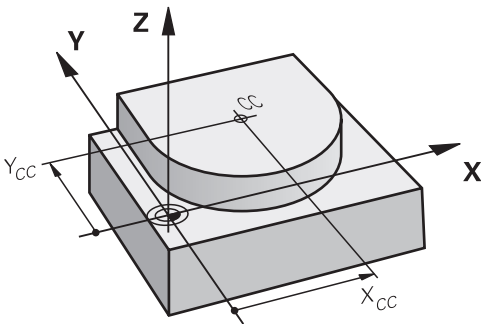
You define a circle center point by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The circle center point remains active until you define a new circle center point. The control does not traverse to the circle center point.

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

> **i** The control simultaneously uses the **CC** function as the pole for polar coordinates.
>
> **Further information:** "Polar coordinate datum at pole CC", Page 241

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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element

Example

5 CC X+25 Y+25

or

10 L X+25 Y+25
11 CC

9.3.6 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 245

Requirement

- Circle center point **CC** is defined
  Further information: "Circle center point CC", Page 229
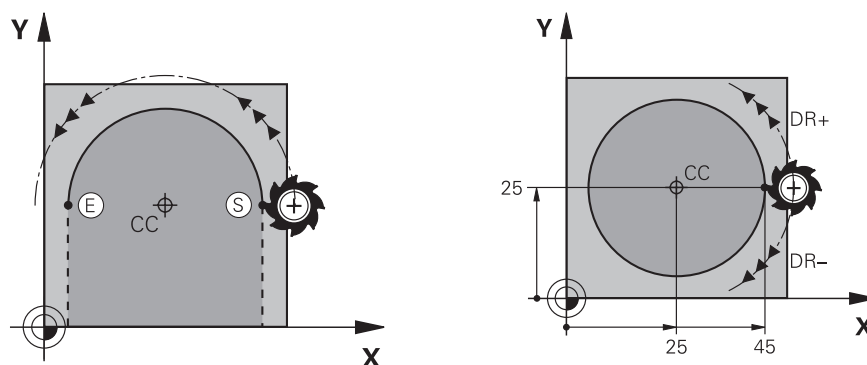
Description of function

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 Number or numerical parameter 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 Number or numerical parameter Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 237 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

Example

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

9.3.7 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.
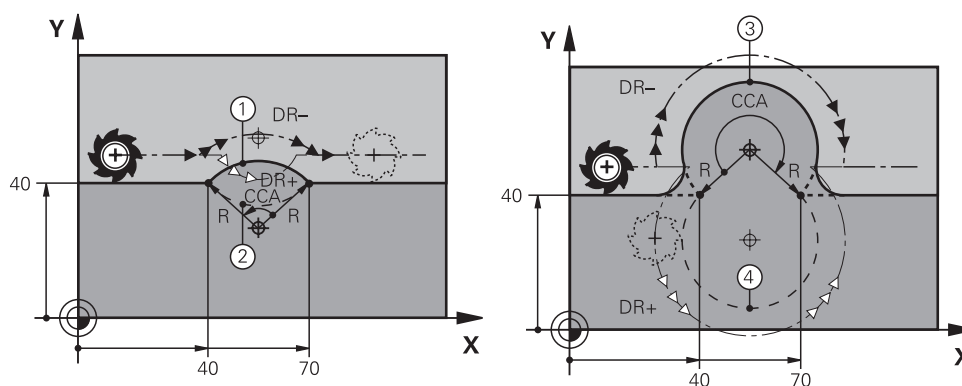
The starting and end points can be connected with four different circular paths of the same radius. The correct circular path is defined with the **CCA** center angle of the circular path radius **R** and the direction of rotation **DR**.

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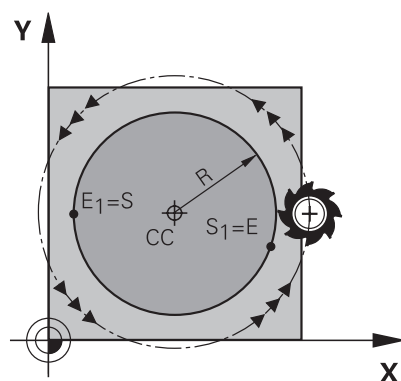
For a full circle, program two circular paths in succession. The end point of the first circular path is the starting point of the second. The end point of the second circular path is the starting point of the first.

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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius of an arc Number or numerical parameter
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 237 Optional syntax element
DR	Rotational direction of the arc Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter Optional syntax element

Note

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

9.3.8 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 247

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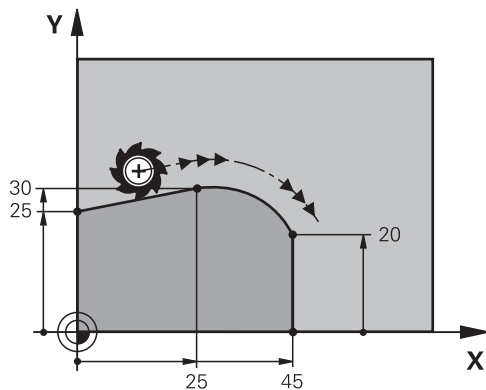
The control moves the tool on a circular path, with a tangential connection, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

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 Number or numerical parameter 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 Number or numerical parameter Entry: absolute or incremental **Further information:** "Linear superimpositioning of a circular path", Page 237 Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.3.9 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 249

Description of function

You can linearly superimpose the following circular paths:

- Circular contour **C**

  Further information: "Circular path C ", Page 230

- Circular contour **CR**

  Further information: "Circular path CR", Page 232

- Circular contour **CT**

  Further information: "Circular path CT", Page 235

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

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

Notes

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

  Further information: "Settings in the Program workspace", Page 139

- 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 224

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 282

9.3.10 Circular path in another plane

Application

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

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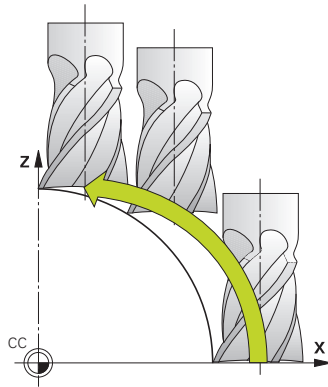
You program circular paths that lie in another plane by entering one axis of the working plane and the tool axis.

Further information: "Designation of the axes of milling machines", Page 124

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+

9.3.11   Example: Cartesian path functions
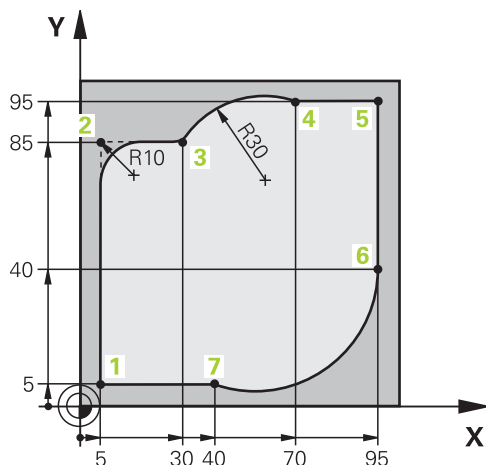
0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Define the workpiece blank for workpiece simulation
3 TOOL CALL 1 Z S4000	; Call the tool in the tool axis and with the spindle speed
4 L Z+250 R0 FMAX	; Retract the tool in the tool axis at rapid traverse FMAX
5 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 of program run
18 END PGM CIRCULAR MM	

9.4 Path functions with polar coordinates

9.4.1 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 243
C	**+**	P	Circular path **CP** (circle polar)	Page 245
			Circular path around circle center point or pole **CC** to arc end point	
CT	**+**	P	Circular path **CTP** (circle tangential polar)	Page 247
			Circular path with tangential connection to the preceding contour element	
C	**+**	P	Helix with circular path **CP** (circle polar)	Page 249
			Combination of a circular and a linear motion	

9.4.2 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 229

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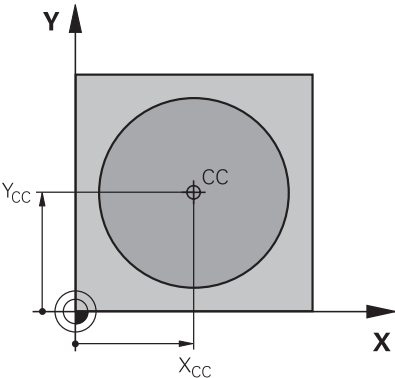
You use the **CC** function to define a position as the pole. You define a pole by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The pole remains active until you define a new pole. The control does not traverse to this position.

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 Number or numerical parameter Entry: absolute or incremental Optional syntax element

Example

11 CC X+30 Y+10

9.4.3   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 224

Requirement

■ Pole **CC**

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

Further information: "Polar coordinate datum at pole CC", Page 241

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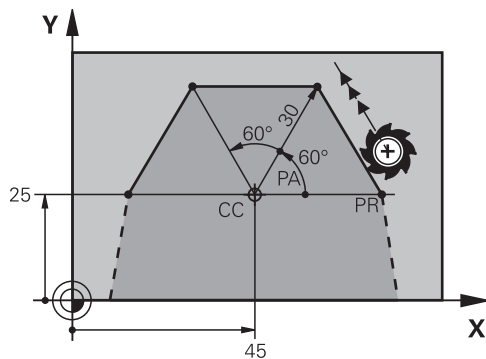
The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

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 ▶ LP

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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 412
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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 147

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

9.4.4 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 230

Requirement

- Pole **CC**

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

    Further information: "Polar coordinate datum at pole CC", Page 241

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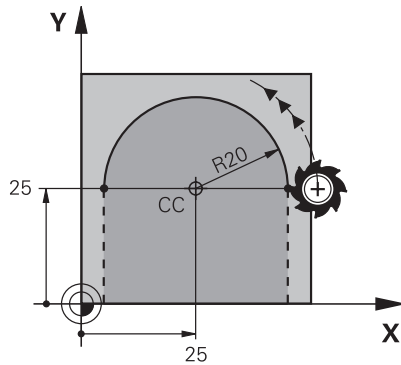
The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block.

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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C,** **U**, **V**, **W**	Axis and value of the linear superimposition
	Number or numerical parameter
	Entry: absolute or incremental
	Further information: "Linear superimpositioning of a circular path", Page 249
	Optional syntax element
DR	Rotational direction of the arc
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 412
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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+

9.4.5 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 235

Requirements

- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 241

- 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
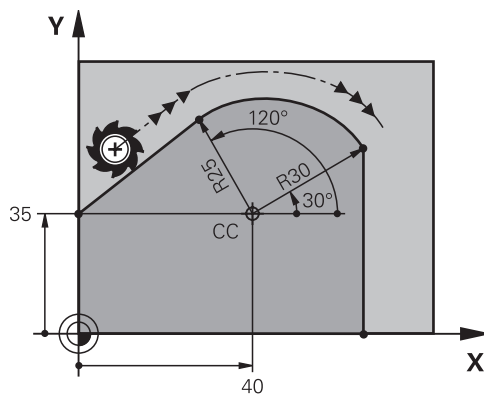
The control moves the tool on a circular path, with a tangential connection, from the current position to the end point defined with polar coordinates. The starting point is the end point of the preceding NC block.

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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
X, **Y**, **Z**, **A**, **B**, **C,** **U**, **V**, **W**	Axis and value of the linear superimposition
	Number or numerical parameter
	Entry: absolute or incremental
	Further information: "Linear superimpositioning of a circular path", Page 249
	Optional syntax element
DR	Rotational direction of the arc
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 412
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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 147

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

9.4.6 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 237

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 245

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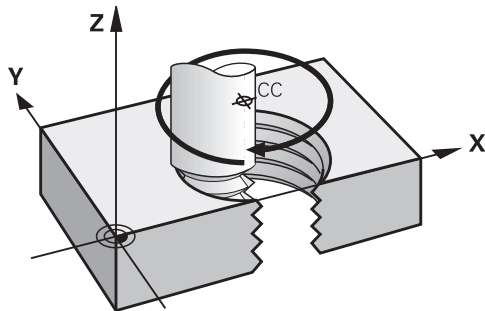
A helix is a combination of a circular path **CP** and a linear motion perpendicular to this path. You program the circular path **CP** in the working plane.

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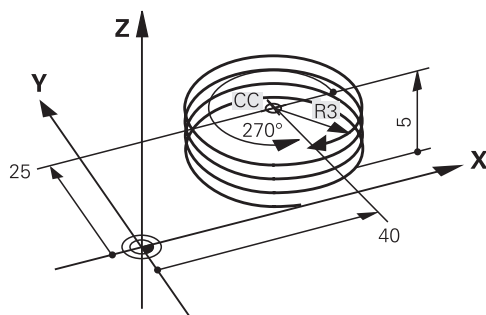
> ⓘ Define the same algebraic sign for the direction of rotation **DR** and the incremental total angle **IPA**. The tool may otherwise move on a wrong path.

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
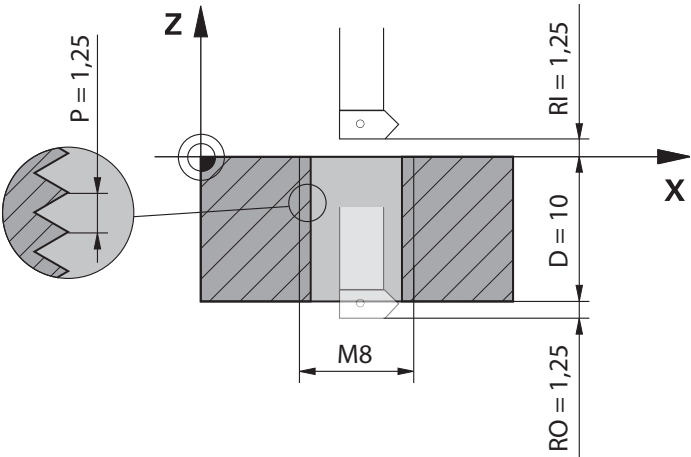
This example includes the following default values:

- **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 250

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 282

Further information: "Example", Page 238

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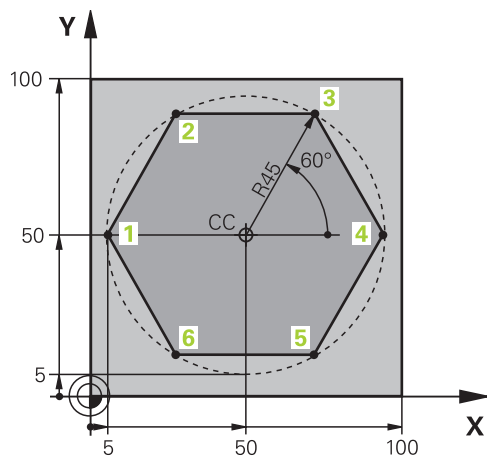
0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Workpiece blank definition
3 TOOL CALL 1 Z S4000	; Tool call
4 CC X+50 Y+50	; Define the datum for polar coordinates
5 L Z+250 R0 FMAX	; Retract the tool
6 LP PR+60 PA+180 R0 FMAX	; Pre-position the tool
7 L Z-5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

9.5 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.

9.5.1 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 256
	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 258
	APPR CT or **APPR PCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection	Page 260
	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 262

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 264
	DEP LN Depart contour on a straight line perpendicular to the last contour point	Page 265
	DEP CT Depart contour on a circular path with a tangential connection	Page 266
	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 266

> ℹ️ 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 216

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 249

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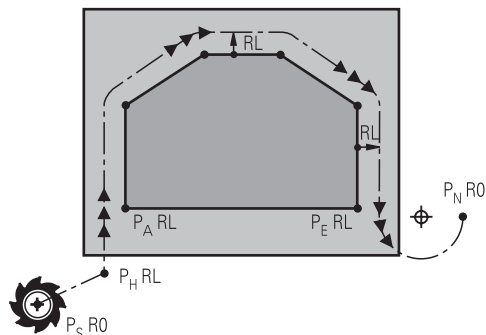
> ### *NOTICE*
>
> **Danger of collision!**
>
> The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **FMAX** in the last positioning block before the approach function, the control also approaches the auxiliary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

9.6   Approach and departure functions with Cartesian coordinates

9.6.1   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 270

Description of function
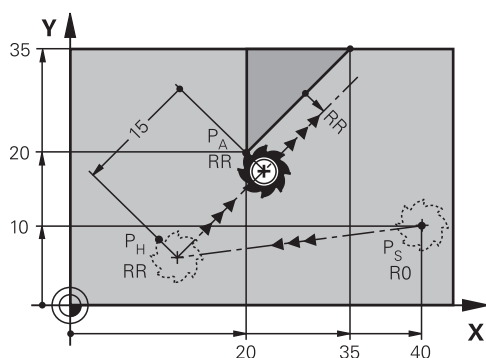
This NC function encompasses the following steps:

- 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** ▶ **Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.6.2   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 272

Description of function
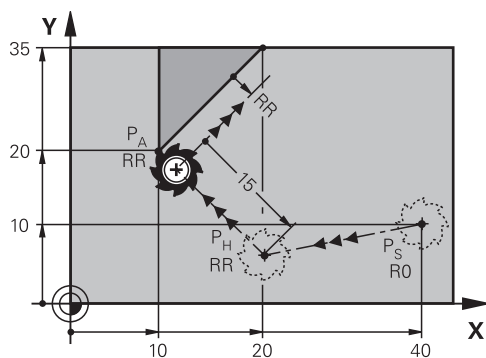
This NC function encompasses the following steps:

■ 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 ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.6.3   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 274

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 ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
CCA	Center angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.6.4   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 277

Description of function
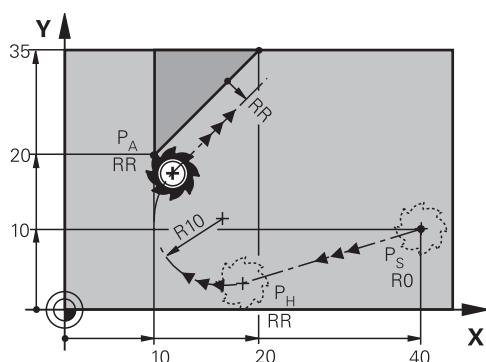
This NC function encompasses the following steps:

- 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 ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.6.5   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
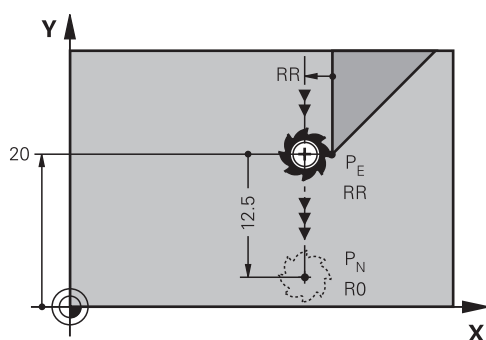
The tool moves in a straight line from the last contour point P_E to the end point P_N.

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 ▶ Departure functions (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 Number or numerical parameter Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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**

9.6.6 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
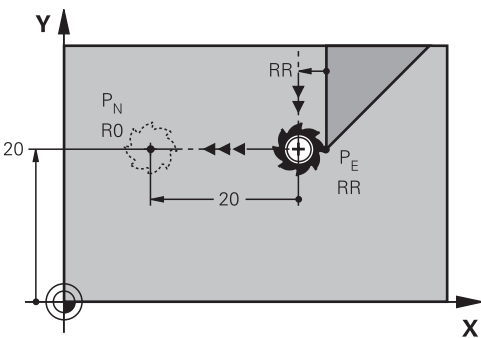
The tool moves in a straight line from the last contour point P_E to the end point P_N.

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 ▶ Departure functions (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
	Number or numerical parameter
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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**

9.6.7   Departure function DEP CT

Application

With the NC function **DEP CT**, the control departs from the contour on a circular path tangential to the last contour element.

Description of function
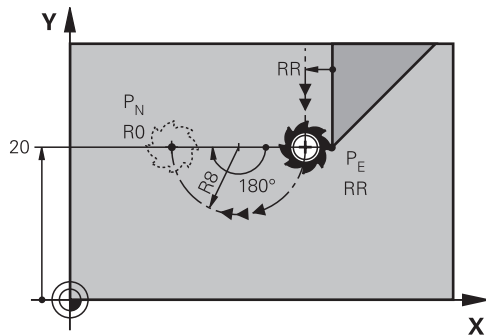
The tool moves on a circular path from the last contour point P_E to the end point P_N.

The circular path is defined by the center angle **CCA** and the radius **R**.

The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> **i** If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 DEP CT CCA30 R+8	; Depart from the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ Departure functions (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
	Number or numerical parameter
R	Radius
	Number or numerical parameter
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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**

9.6.8 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 279

Description of function
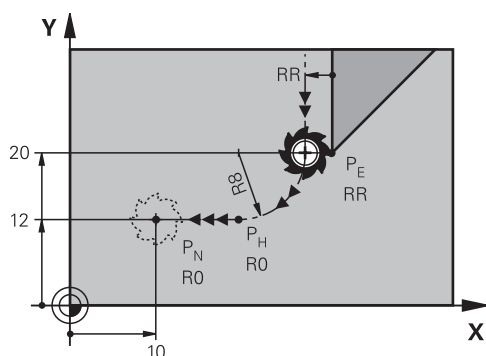
This NC function encompasses the following steps:

- 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** ▶ **Departure functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius Number or numerical parameter
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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**

9.7   Approach and departure functions with polar coordinates

9.7.1   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 256

Requirement

- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 241

Description of function
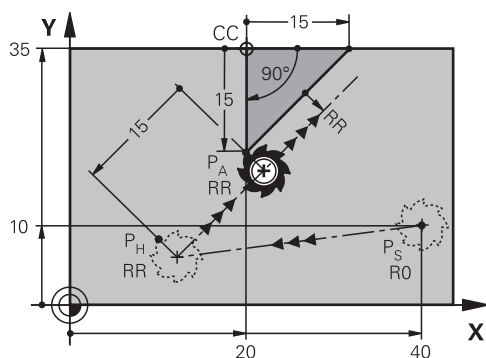
This NC function encompasses the following steps:

- 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 ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.7.2   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 258

Requirement

■ Pole **CC**

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

   Further information: "Polar coordinate datum at pole CC", Page 241

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 ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
LEN	Distance of the auxiliary point P_H to the contour Number or numerical parameter Entry: absolute or incremental Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.7.3 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 260

Requirement

■ Pole **CC**

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

Further information: "Polar coordinate datum at pole CC", Page 241

Description of function

This NC function encompasses the following steps:

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

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

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

  The circular path is defined by the center angle **CCA** and the radius **R**.

  The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of the radius **R**.

The table shows the relationship between tool radius compensation and the algebraic sign of the radius **R** and the direction or rotation:

Radius compensation	Algebraic sign of radius	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

> **i** If you change the algebraic sign of the radius **R**, then the position of the auxiliary point P_H changes.

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Input

11 APPR PCT PR+15 PA-90 CCA180 R +10 RL F300	; Approach the contour on a tangential circular path

To navigate to this function:

Insert NC function ▶ All functions ▶ Path contour ▶ Approach functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
CCA	Center angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius Number or numerical parameter Optional syntax element
R0, **RL**, **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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

9.7.4 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 262

Requirement

- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 241

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 ▶ Approach functions (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
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
PA	Polar coordinate angle
	Number or numerical parameter
	Entry: absolute or incremental
	Optional syntax element
R	Radius
	Number or numerical parameter
	Optional syntax element
R0, **RL**, **RR**	Tool radius compensation
	Further information: "Tool radius compensation", Page 412
	Optional syntax element
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate
	Further information: "Feed rate F", Page 211
	Number or numerical parameter
	Optional syntax element
M	M function
	Further information: "Miscellaneous Functions", Page 553
	Number or numerical parameter
	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 147

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

9.7.5 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 267

Requirement

- Pole **CC**

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

  Further information: "Polar coordinate datum at pole CC", Page 241

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 ▶ Departure functions (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 Number or numerical parameter Entry: absolute or incremental Optional syntax element
PA	Polar coordinate angle Number or numerical parameter Entry: absolute or incremental Optional syntax element
R	Radius Number or numerical parameter
F, **FMAX**, **FZ**, **FU**, **FAUTO**	Feed rate **Further information:** "Feed rate F", Page 211 Number or numerical parameter Optional syntax element
M	M function **Further information:** "Miscellaneous Functions", Page 553 Number or numerical parameter 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 147

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**

10

Programming techniques

10.1 Subprograms and program section repeats with the label LBL

Application

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary. Use subprograms to insert contours or complete machining steps after the end of the program and call them in the NC program. Program section repeats repeat single or several NC blocks during the NC program. Subprograms and program section repeats can also be combined.

Subprograms and program section repeats are programmed with the NC function **LBL**.

Related topics

- Executing NC programs within another NC program

  Further information: "Calling an NC program with CALL PGM", Page 300

- Jumps with conditions as if-then decisions.

  Further information: "The Jump commands folder", Page 619

Description of function

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

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

Key or icon	Function
LBL SET	Create **LBL**
LBL CALL	Call **LBL**: Jump to the label in the NC program
↻ ¹₂...	In case of **LBL** number: Enter the next free number automatically

Defining a label with LBL SET

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

Each label must be unambiguously identifiable in the NC program by its number or name. If a number or a name exists twice in an NC program, the control shows a warning before the NC block.

LBL 0 marks the end of a subprogram. This number is the only one which may exist more than once in the NC program.

Input

11 LBL "Reset"	; Subprogram for resetting a coordinate transformation
12 TRANS DATUM RESET	
13 LBL 0	

To navigate to this function:

Insert NC function ▶ All functions ▶ Label ▶ LBL SET

The NC function includes the following syntax elements:

Syntax element	Meaning
LBL	Syntax initiator for a label
Number or **Name**	Number or name of the label
	Number, text, or variable
	Input: **0...65535** or **text width 32**
	Use an icon to enter the next free number automatically.
	Further information: "Description of function", Page 282

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 **Parameter**	Number or name of the label
	Number, text, or variable
	Input: **1...65535** or **text width 32** or **0...1999**
	The label can be selected from a selection menu that displays all labels available in the NC program.
REP	Number of repetitions until the control executes the next NC block
	Optional syntax element

Subprograms

```
 0   BEGIN PGM ...
     ...        ①
11   CALL LBL 1 ───────┐
     ...        ⑤   ←──┐│
                       ②│
20   M30               ││
21   LBL 1   ←─────────┘│
     ...        ③        │
                        ④
25   LBL 0   ────────────┘
26   END PGM ...
```
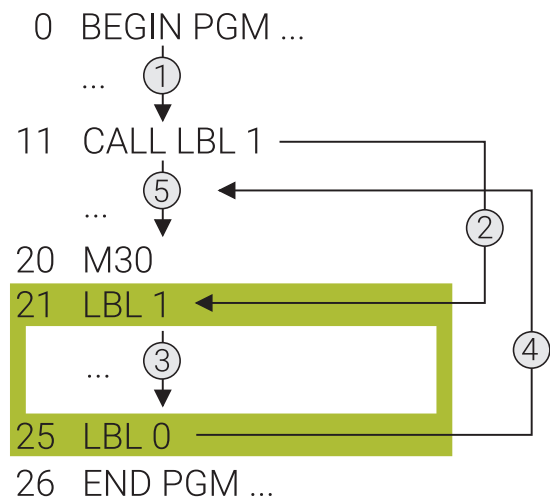
A subprogram allows calling parts of an NC program any number of times at different points of the NC program (e.g., machining positions or a contour).

A subprogram starts with a **LBL** label and ends with **LBL 0**. **CALL LBL** calls the subprogram from any point in the NC program. In this process, repetitions must not be defined with **REP**.

The control executes the NC program as follows:

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

2   The control jumps to the beginning of the defined subprogram **LBL**.

3   The control executes the subprogram up to the subprogram end **LBL 0**.

4   After that, the control jumps to the next NC block after **CALL LBL** and continues executing the NC program.

The following conditions apply to subprograms:

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

    If subprograms are located in the NC program before the NC block with M2 or M30, they will be executed at least once even if they are not called

The control displays information about the active subprogram on the **LBL** tab of the **Status** workspace.

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

Program-section repeats

```
 0  BEGIN PGM ...
    ...        ①

11  LBL 1
         ②        ④           ⑥
              REP 1        REP 2
    ...      ③        ⑤

21  CALL LBL 1 REP2
    ...       ⑦

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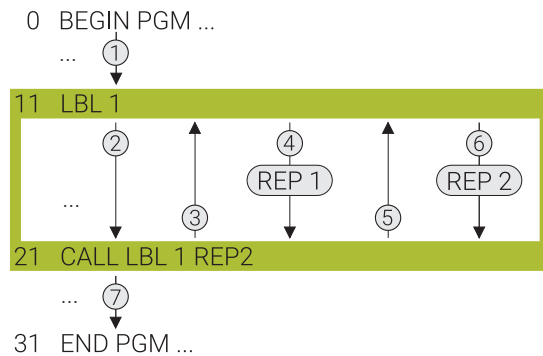
A program section repeat allows repeating a part of an NC program any number of times (e.g., contour machining with incremental infeed).

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

The control executes the NC program as follows:

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

  In this process, the control already executes the program section once because the program section to be repeated is positioned ahead of the **CALL LBL** function.

2 The control jumps to the beginning of the program section repeat **LBL**.

3 The control repeats the program section as many times as programmed under **REP**.

4 After that, the control continues executing the NC program.

The following conditions apply to program section repeats:

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

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

Further information: 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 737
- 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> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

10.2   Nesting with LBL

Application

You can combine program section repeats and subprograms with each other and also with other programming techniques, such as calling a separate NC program.

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 284

■ Program section repeats

Further information: "Program-section repeats", Page 285

■ Calling a separate NC program

Further information: "Selection functions", Page 300

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

10.2.1 Example

Subprogram call within a subprogram

0 BEGIN PGM UPGMS MM	
* - …	
11 CALL LBL "UP1"	; Call subprogram **"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 **2**
* - …	
41 LBL 0	; End of subprogram **"UP1"**
42 LBL 2	; Start of subprogram **2**
* - …	
51 LBL 0	; End of subprogram **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.

10.3   Control structures

10.3.1   Fundamentals

The control provides the NC functions for programming the control structures.

Using control structures, you can program the NC program more clearly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends.

The control provides the following NC functions:

NC function	Meaning
Case analyses: ■ **IF** ■ **ELSE IF** ■ **ELSE**	Case analyses allow you to define conditions. If a condition is fulfilled, the control executes the NC blocks that belong to that condition. If a condition is not fulfilled, the control skips the NC blocks. **Further information:** "Case analyses", Page 289
Program loops: ■ **FOR** ■ **WHILE**	Program loops enable you to repeat NC blocks several times. **Further information:** "Program loops", Page 292
Advanced control of program loops: ■ **BREAK** ■ **CONTINUE**	The NC functions for advanced control of program loops allow you to skip the current loop repetition or cancel the program loop. You can program these NC functions within **FOR** and **WHILE** loops. **Further information:** "Advanced controlling of program loops", Page 296

10.3.2   Case analyses

Application

You can use the **IF**, **ELSE IF** and **ELSE** case analyses to control the actions of the NC program. You use conditions to define whether the control executes specific program sections or skips them.

Related topics

■ Jump commands with **FN**

   Further information: "The Jump commands folder", Page 619

Description of function

You use the **IF** and **END IF** NC functions to define the header and footer of a case analysis. In the header you define a condition.

If the condition is fulfilled, the control executes all NC blocks located between the header and footer.

The control indents the NC blocks between the header and footer.

Within the case analysis you can use **ELSE IF** to define further conditions, and **ELSE** to define an alternative program section. The control executes the alternative program section only if none of the previous conditions are fulfilled.

You can use numerical values when programming the following conditions:

Condition			Meaning
a	==	b	a equal to b
a	!=	b	a not equal to b
a	<	b	a less than b
a	>	b	a greater than b
a	<=	b	a less than or equal to b
a	>=	b	a greater than or equal to b

You can use alpha-numerical values when programming the following conditions:

Condition			Meaning
a	==	b	a equal to b
a	!=	b	a not equal to b
a	IN	b	a contained in b
			The control checks whether the content to the left of **IN** is contained in the same sequence in the content to the right. The control also takes capitalization into account.

Input

IF

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	; If this condition is fulfilled, the indented content is executed
21 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Sorting tests ▶ IF and END IF

The NC function includes the following syntax elements:

Syntax element	Meaning
IF	Syntax initiator for case analysis with **IF**
Q50 < Q60	Condition Number, text, or variable

> **i** You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

ELSE IF

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	
21 ELSE IF Q50 > Q60	; Header with additional condition
* - ...	; If this condition is fulfilled, the indented content is executed
31 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Sorting tests ▶ ELSE IF

The NC function includes the following syntax elements:

Syntax element	Meaning
ELSE IF	Syntax initiator for another condition with **ELSE IF**
Q50 > Q60	Condition
	Number, text, or variable

ELSE

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	
21 ELSE	; Header for alternative execution
* - ...	; If the previous condition is not fulfilled, the indented content is executed
31 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Sorting tests ▶ ELSE

The NC function includes the following syntax elements:

Syntax element	Meaning
ELSE	Syntax initiator for alternative execution with **ELSE**

Notes

- If you program numerical parameters in the condition, then you must define the variables before the case analysis.
- The control always processes only one case per level. If, for example, the **IF** condition is fulfilled, the control skips all other cases on the same level.
- If multiple conditions should need to be fulfilled for a program section, then you must nest the case analysis. You can use nesting, for example, to program that a value should be within a numerical range.

  Further information: "Example", Page 298
- If you program jump commands (e.g., using **FN 9**) within case analyses, the control will display an error message.

  Further information: "The Jump commands folder", Page 619
- Please note that the control considers digits from the tenth decimal place onward to be identical. This means, for example, that calculation results differing only after the ninth decimal place will be considered to have the same value.

10.3.3   Program loops

FOR loop

Application

You can use a **FOR** loop to program simple program-section repeats.

Related topics

- Subprograms and program-section repeats

  Further information: "Subprograms and program section repeats with the label LBL", Page 282

Description of function

A program loop consists of a header and footer and all the defined NC blocks between them. You use the **FOR** and **END FOR** NC functions to define the header and footer.

The control indents the NC blocks between the header and footer.
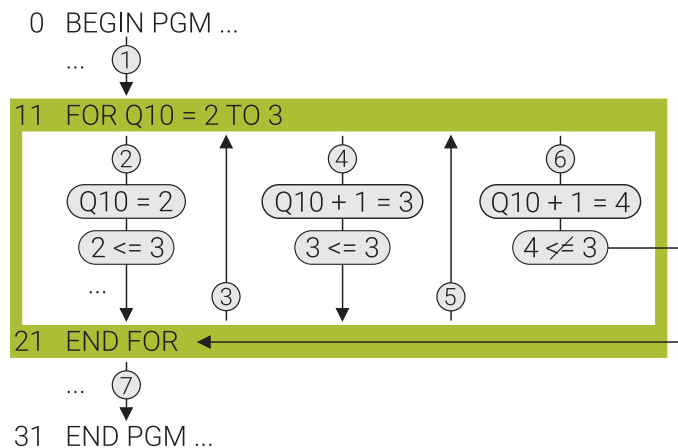
The control executes the NC program as follows:

1   The control reads the header with the counting variable, the target value, and the increment, if given.

2   The control writes the starting value to the counting variable.

    Since the value of the counting variable is less than or equal to the target value, the control executes the loop content.

3   The control reads the footer and returns to the header.

4   The control changes the value of the counting variable by the increment.

    Since the value of the counting variable is less than or equal to the target value, the control executes the loop content.

5   The control reads the footer and returns to the header.

6   The control changes the value of the counting variable by the increment.

    Since the value of the counting variable is greater than the target value, the control skips the loop content and jumps directly to the footer.

7   The control ends the program loop and continues the NC program.

Input

11 FOR Q50 = 4 TO 10 STEP 2	; Header of the **FOR** loop
* - ...	; Content is executed until the value of **Q50** is greater than **10**
21 END FOR	; Footer of the **FOR** loop

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Program section repeat ▶ FOR header and footer

The NC function includes the following syntax elements:

Syntax element	Meaning
FOR	Syntax initiator of a **FOR** loop
Q50	Counting variable
	With each repetition of the program loop the control checks the value of this variable. If the value is less than the target value, the control changes this value by the increment.
= 4	Start value
	At the start of the program loop, the control assigns this value to the counting variable once.
	Only integers smaller than the target value are possible.
	Number or numerical parameter
TO 10	Target value
	The control repeats the program loop until the value of the counting variable is greater than the target value.
	Only integers are possible
	Number or numerical parameter
STEP 2	Increment
	Only integers are possible
	If you don't define an increment, the control uses the value **1**.
	Number or numerical parameter
	Optional syntax element

> ⓘ You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

Notes

- If you program numerical parameters in the header, then you must define the variables before the program loop.
- Do not assign a new value to the counting variable within the program loop, since that could lead to an infinite loop.
- If you program jump commands (e.g., using **FN 9**) within program loops, the control will display an error message.
  Further information: "The Jump commands folder", Page 619

WHILE loop

Application

You can use a **WHILE** loop to program simple program-section repeats with various conditions. The conditions can contain numerical or alpha-numerical values.

Related topics

- Subprograms and program-section repeats

  Further information: "Subprograms and program section repeats with the label LBL", Page 282

Description of function

A program loop consists of a header and footer and all the defined NC blocks between them. You use the **WHILE** and **END WHILE** NC functions to define the header and footer.

The control indents the NC blocks between the header and footer.

```
0   BEGIN PGM ...
    ...         ①
5   Q10 = 2
    ...
11  WHILE Q10 < 4
          ②              ④              ⑥
      (Q10 = 2)      (Q10 = 3)      (Q10 = 4)
       (2 < 4)        (3 < 4)        (4 ≮ 4)
15      Q10 = Q10 + 1
    ...     ③          ⑤
21  END WHILE
    ...     ⑦
31  END PGM ...
```
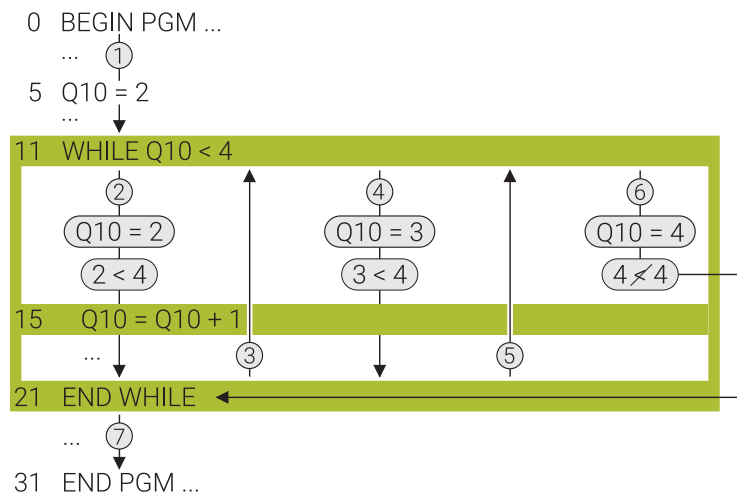
The control executes the NC program as follows:

1. The control assigns the value **2** to the variable **Q10** and then reads the header with the condition.
2. The control checks whether the condition is fulfilled.
   Since the condition is fulfilled, the control executes the loop content.
   Within the program loop the control increments the value of the variable **Q10**.
3. The control reads the footer and returns to the header.
4. The control checks whether the condition is fulfilled.
   Since the condition is fulfilled, the control executes the loop content.
   Within the program loop the control increments the value of the variable **Q10**.
5. The control reads the footer and returns to the header.
6. The control checks whether the condition is fulfilled.
   Since the condition is no longer fulfilled, the control skips the loop content and jumps directly to the footer.
7. The control ends the program loop and continues the NC program.

You can use numerical values when programming the following conditions:

Condition			Meaning
a	**==**	b	a equal to b
a	**!=**	b	a not equal to b
a	**<**	b	a less than b
a	**>**	b	a greater than b
a	**<=**	b	a less than or equal to b
a	**>=**	b	a greater than or equal to b

You can use alpha-numerical values when programming the following conditions:

Condition			Meaning
a	**==**	b	a equal to b
a	**!=**	b	a not equal to b
a	**IN**	b	a contained in b
			The control checks whether the content to the left of **IN** is contained in the same sequence in the content to the right. The control also takes capitalization into account.

Input

11 Q50 = +5	; Define **Q50** with the value **5**
12 Q60 = +10	; Define **Q60** with the value **10**
13 WHILE Q50 <= Q60	; Header of the **WHILE** loop
14 Q50 = Q50 + +1	; Increment **Q50** by the value **1**
* - ...	; Content will be executed until the value in **Q50** is greater than the value in **Q60**
21 END WHILE	; Footer of the **WHILE** loop

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Program section repeat ▶ WHILE header and footer

The NC function includes the following syntax elements:

Syntax element	Meaning
WHILE	Syntax initiator of a **WHILE** loop
Q50 <= Q60	Condition Number, text, or variable

> ℹ️ You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

Notes

- If you program numerical parameters in the header, then you must define the variables before the program loop.
- You can use a **WHILE** loop to achieve the functionality of a **FOR** loop.. To do so, define a calculation step within the program loop in order to increment or decrement a value.
- If you program jump commands (e.g., using **FN 9**) within program loops, the control will display an error message.
  Further information: "The Jump commands folder", Page 619
- Please note that the control considers digits from the tenth decimal place onward to be identical. This means, for example, that calculation results differing only after the ninth decimal place will be considered to have the same value.

10.3.4 Advanced controlling of program loops

Aborting program loops with BREAK

Application

You can use the **BREAK** NC function to abort a program loop (e.g., if a certain machine or tool status occurs).

Description of function

If the control executes the **BREAK** NC function, it aborts the program loop. The control skips the NC blocks until the footer and then executes the rest of the NC program.

Input

11 FOR Q50 = 4 TO 10	; Header of the **FOR** loop
* - …	
15 IF Q182 == +1	; Workpiece status: scrap after probing
16 BREAK	; If **Q182** contains the value **1**, the control will execute this NC block contains and then abort the program loop
17 END IF	
* - …	
21 END FOR	; Footer of the **FOR** loop

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Program section repeat ▶ BREAK

The NC function includes the following syntax elements:

Syntax element	Meaning
BREAK	Syntax initiator for aborting the program loop

Note

Program a **BREAK** only in combination with a case analysis. That way the control will abort the program loop only in the defined case.
Further information: "Case analyses", Page 289

Canceling program loops with CONTINUE

Application

You can use the **CONTINUE** NC function to skip the current repetition of a program loop. The control then continues with the subsequent program loop.

Description of function

If the control executes the **CONTINUE** NC function, it cancels the current loop repetition. The control skips the NC blocks up to the footer, ignoring the rest of the program loop. The control then executes the next loop repetition.

Input

11 FOR Q50 = 4 TO 10	; Header of the **FOR** loop
* - ...	
15 IF Q182 == +1	; Workpiece status: scrap after probing
16 CONTINUE	; If **Q182** contains the value **1**, the control will execute this NC block contains and then cancel the current loop repetition
17 END IF	
* - ...	
21 END FOR	; Footer of the **FOR** loop

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Program section repeat ▶ CONTINUE

The NC function includes the following syntax elements:

Syntax element	Meaning
CONTINUE	Syntax initiator for canceling a loop repetition

10.3.5 Example

This example combines various control structures with each other, and also uses nesting:

0 BEGIN PGM EXAMPLE MM	
1 * - ...	; The purpose of this program is to finish the milling of hardened workpieces
2 Q1400 = +19.995	; Lower tolerance
3 Q1401 = +20	; Nominal dimension
4 Q1402 = +20.005	; Upper tolerance
5 BLK FORM 0.1 Z X-50 Y-50 Z-20	
6 BLK FORM 0.2 X+50 Y+50 Z+0	
7 CALL LBL "measure"	; Measure circular pocket
8 IF Q966 < Q1400	; Check whether circular pocket is smaller than the lower tolerance
9 WHILE Q966 < Q1400	; Program loop for reworking the circular pocket
10 CALL LBL "rework"	; Mill the circular pocket
11 CALL LBL "measure"	; Measure circular pocket
12 END WHILE	; End of the program loop
13 ELSE	; If the condition in NC block **8** is not fulfilled, the circular pocket is larger than the lower tolerance
14 IF Q966 <= Q1402	; Check whether circular pocket is larger than the upper tolerance
15 FN 16: F-PRINT pass.a / SCREEN:	; Output on screen: Circular pocket is OK
16 ELSE	; If the condition in NC block **14** is not fulfilled, the circular pocket is larger than the upper tolerance
17 FN 16: F-PRINT scrap.a / SCREEN:	; Output on screen: Circular pocket is too large
18 END IF	
19 END IF	
20 L Z+100 R0 FMAX	
21 M30	
22 LBL "measure"	
23 TOOL CALL 600 Z	; Call touch probe
24 L Z+100 R0 FMAX	
25 TCH PROBE 1401 CIRCLE PROBING ~	
Q1100=+0 ;1ST POINT REF AXIS ~	
Q1101=+0 ;1ST POINT MINOR AXIS ~	
Q1102=-5 ;1ST POINT TOOL AXIS ~	
Q1116=+20 ;DIAMETER 1 ~	
Q1115=+0 ;GEOMETRY TYPE ~	
Q423=+4 ;NO. OF PROBE POINTS ~	
Q325=+0 ;STARTING ANGLE ~	
Q1119=+360 ;ANGULAR LENGTH ~	
Q320=+0 ;SET-UP CLEARANCE ~	
Q260=+100 ;CLEARANCE HEIGHT ~	

Q1125=+2 ;CLEAR. HEIGHT MODE ~	
Q309=+0 ;ERROR REACTION ~	
Q1120=+0 ;TRANSFER POSITION	
26 * - ...	; The control stores the diameter in **Q966**
27 LBL 0	
28 LBL "rework"	
29 Q1500 = (Q1401 - Q966) / +2	; Calculate tool compensation from the nominal dimension and the measured diameter
30 TOOL CALL 5 Z S2000 F200 DR-Q1500	; Call milling tool
31 L Z+100 R0 FMAX M3	
32 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+2 ;MACHINING OPERATION ~	
Q223=+20 ;CIRCLE DIAMETER ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q207=AUTO ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-20 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q366=+1 ;PLUNGE ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	
33 L X+0 Y+0 R0 FMAX M99	
34 LBL 0	
35 END PGM EXAMPLE MM	

10.4   Selection functions

10.4.1   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 300
⊕	Select a datum table with **SEL TABLE**	Page 332
░░░	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 302
📄	Call the last selected file with **CALL SELECTED PGM**	Page 302
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 418
🗁	Open the file with **OPEN FILE**	Page 462
⬠	Link multiple contours with **CONTOUR DEF**	

10.4.2   Calling an 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 302

- Executing multiple NC programs as a job list

  Further information: "Pallet machining and job lists", Page 787

Description of function

```
 0   BEGIN PGM A                    0   BEGIN PGM B
     ...  ①          ②
11   CALL PGM B                         ...
     ...  ⑤
                     ④
21   END PGM A                      21   END PGM B
```
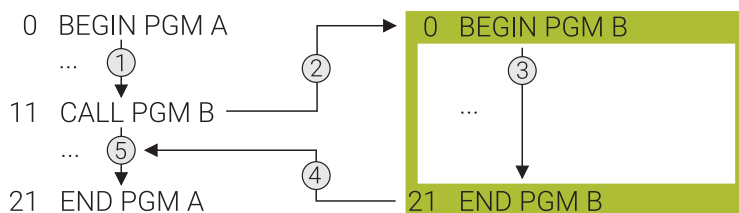
The control executes the NC program as follows:

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 620

  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 302
- As a rule, variables are globally effective when called, such as with **CALL PGM**. Please note that changes to variables in the called NC program can also take effect on the calling NC program. If applicable, use QL or named 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.

10.4.3 Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM

Application

The function **SEL PGM** allows selecting another separate NC program that you can call at a different position in the active NC program. The control executes the selected NC program at the position where you call it in the calling NC program using **CALL SELECTED PGM**.

Related topics
- Calling the NC program directly
  Further information: "Calling an NC program with CALL PGM", Page 300

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 620

  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 **Parameter**	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 629

- 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, variables are globally effective when called, such as with **CALL PGM**. Please note that changes to variables in the called NC program can also take effect on the calling NC program. If applicable, use QL or named 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.

10.5 NC sequences for reuse

Application

You can save up to 2000 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 148
- Mark and copy NC blocks with the context menu
  Further information: "Context menu", Page 747
- Call NC programs unchanged
  Further information: "Calling an NC program with CALL PGM", Page 300

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 750
- 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

> **i** You must use the **Create NC sequence** button to create the first NC sequence; then the **TNC:\system\PGM-Templates** folder is created.

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.
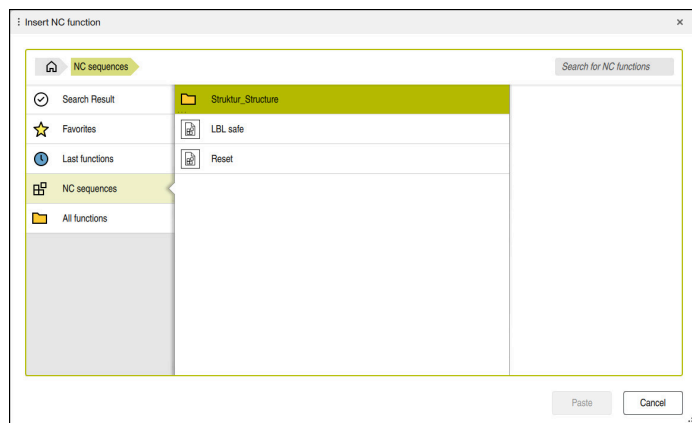
Inserting NC sequences in the **Insert NC function** window

If you open an NC sequence as its own tab in the **Editor**, then you can permanently edit the contents of the NC sequence.

Adding a user-defined folder icon

You can save user-defined folder icons for the subfolders of the NC components.

To save a folder icon:

► Select the **Files** operating mode
► Open a subfolder in the **TNC:\system\PGM-Templates** folder
► Insert the desired folder icon with the name **foldericon.svg**
> The control displays the user-defined folder icon in the **NC sequences** area of the **Insert NC function** window.

> ⓘ   You can add a different folder icon with the name **foldericon-dark.svg** for the Dark Mode.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If you insert an NC sequence with a different unit of measure despite the confirmation request, the control will not convert the values. Be aware that input values with a different unit of measure can be too small or too high, such as for feed rates. Incorrect input values can lead to increased tool wear, tool breakage, and lower surface quality.

▶ Always insert NC sequences only into NC programs with the same unit of measure.

▶ Check the input values and correct them if required

■ 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

You can also select some of these file functions from the context menu.

Further information: "Context menu in the Insert NC function window", Page 751

■ 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

■ The input ranges for millimeter values in NC programs are greater than for inch values. The control will display an error message if the millimeter values of an NC sequence exceed the input ranges of an inch program.

■ The control scales user-defined folder icons to max. 32x32 dp (device-independent pixels).

11

Coordinate transfor-
mation

11.1 Reference systems

11.1.1 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:

Abbrevia-tion	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 312
B-CS	Basic coordinate system basic coordinate system	Page 315
W-CS	Workpiece coordinate system workpiece coordinate system	Page 316
WPL-CS	Working plane coordinate system working plane coordinate system	Page 318
I-CS	Input coordinate system input coordinate system	Page 321
T-CS	Tool coordinate system tool coordinate system	Page 323

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.

11.1.2 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.
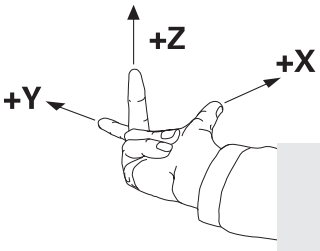
Origin of the coordinate system

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**.
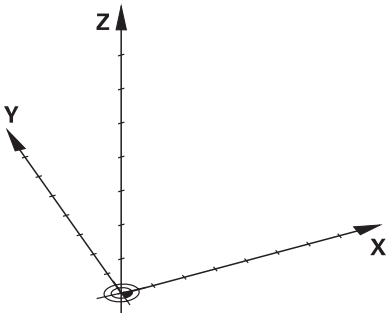
11.1.3 Machine coordinate system M-CS

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.
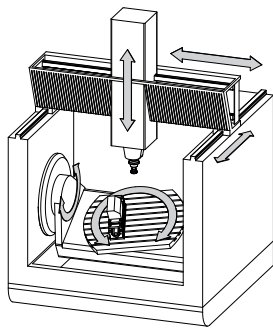
The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

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.
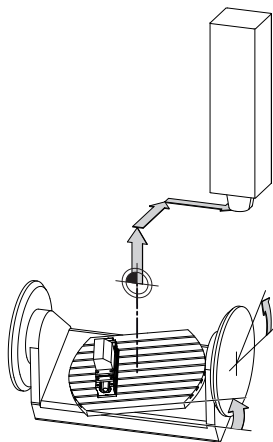
Position of the machine datum in the machine

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 331

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

  Further information: "Datum shift with TRANS DATUM", Page 334

- **Additive offset (M-CS)** function for rotary axes in the **GPS** (#44 / #1-06-1) workspace

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

  > ⚙ The machine manufacturer can also define further transformations.
  > **Further information:** "Note", Page 314

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 558

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 804

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**.

11.1.4 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
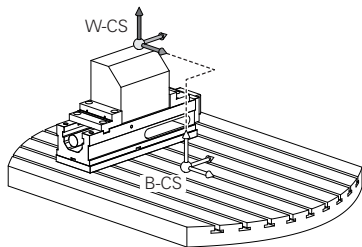
⚙ The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset table in accordance with the machine.

Further information: "Note", Page 316

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

---

11.1.5 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
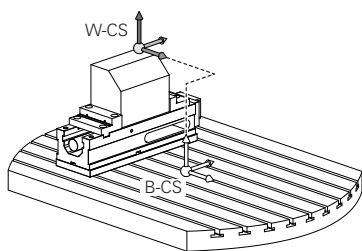
Transformations in the workpiece coordinate system (W-CS)

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 334
■ Columns **X**, **Y**, **Z** of the datum table before tilting the working plane

  Further information: "Datum table", Page 331
■ The **TRANS MIRROR** function or Cycle **8 MIRRORING** before tilting the working plane with spatial angles

  Further information: "Mirroring with TRANS MIRROR", Page 336

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 348

> **ⓘ** You can still execute 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.
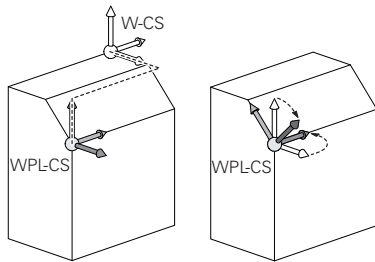
NOTICE

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.

Further information: "Difference between spatial angles and axis angles", Page 345

Additional transformations with Global Program Settings (GPS (#44 / #1-06-1))

In the **GPS** workspace (#44 / #1-06-1), you can define the following additional transformations in the workpiece coordinate system **W-CS**:

- **Additive basic rotat. (W-CS)**

  The effects of this function are added to a basic rotation or a 3D basic rotation from the preset table or the pallet preset table. This function is the first transformation that is possible in the **W-CS**.

- **Shift (W-CS)**

  This function is in effect in addition to a datum shift defined in the NC program with the **TRANS DATUM** function and before the working plane is tilted.

- **Mirroring (W-CS)**

  The function is in effect in addition to a mirror image (**TRANS MIRROR** function or Cycle **8 MIRRORING**) defined in the NC program and before tilting the working plane.

- **Shift (mW-CS)**

  This function is in effect in the modified workpiece coordinate system. This function is active after the **Shift (W-CS)** and **Mirroring (W-CS)** functions and before the working plane is tilted.

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

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 321

- 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 318

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

11.1.6 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 321

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 316

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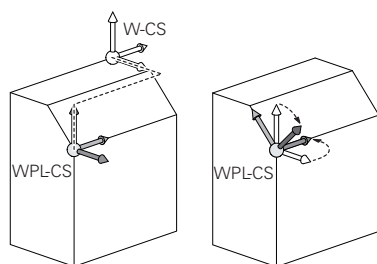
Transformations in the working plane coordinate system WPL-CS

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 334
- The **TRANS MIRROR** function or Cycle **8 MIRRORING**

  Further information: "Mirroring with TRANS MIRROR", Page 336
  Further information: User's Manual for Machining Cycles
- The **TRANS ROTATION** function or cycle **10 ROTATION**

  Further information: "Rotations with TRANS ROTATION", Page 338
  Further information: User's Manual for Machining Cycles
- The **TRANS SCALE** function or cycle **11 SCALING FACTOR**

  Further information: "Scaling with TRANS SCALE", Page 340
  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 373

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.
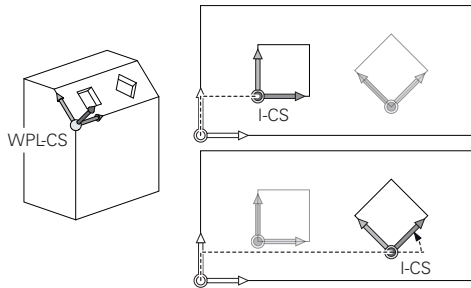
NOTICE

Danger of collision!

The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.

- ▶ Program only the recommended transformations in the respective reference system
- ▶ Use tilting functions with spatial angles instead of with axis angles
- ▶ Use the Simulation mode to test the NC program

Additional transformations with Global Program Settings (GPS (#167 / #1-02-1))

The **Rotation (WPL-CS)** transformation in the **GPS** workspace has an additive effect to a rotation in the NC program.

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

Additional transformations with Mill Turning (#50 / #4-03-1)

With the Turning software option, the following additional transformations are available:

- Precession angle with the following cycles:
  - Cycle **800 ADJUST XZ SYSTEM**
  - Cycle **801 RESET ROTARY COORDINATE SYSTEM**
  - Cycle **880 GEAR HOBBING**
- OEM transformations defined by machine manufacturers for special turning kinematics

> **i** Machine manufacturers can also define an OEM transformation and a precession angle without the Turning (#50 / #4-03-1) software option.
>
> An OEM transformation takes effect before the precession angle.
>
> If an OEM transformation or a precession angle is defined, the control shows the values on the **POS** tab of the **Status** workspace. These transformations are also in effect in milling mode!
>
> **Further information:** User's Manual for Setup and Program Run

Additional transformation with Gear Cutting (#157 / #4-05-1)

You can use the following cycles to define a precession angle:

- Cycle **286 GEAR HOBBING**
- Cycle **287 GEAR SKIVING**

> **i** Even without the Gear Cutting (#157 / #4-05-1) software option, the machine manufacturer can define a precession angle.

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 321

- 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**.

11.1.7 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 318

If no transformations are defined in the **WPL-CS**, then the position and orientation of the **WPL-CS** and **I-CS** are identical.
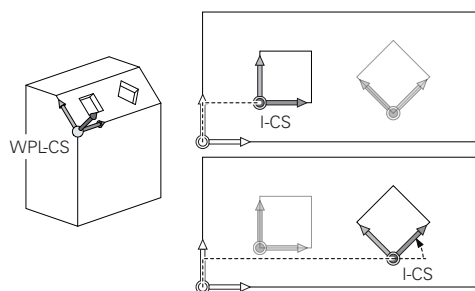
Positioning blocks in the input coordinate system I-CS

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 323

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 318

11.1.8 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.

By making entries in the tool management, you can 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 126

Use the following tool management parameters to define the tool tip relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (#50 / #4-03-1)
- **XL** (#50 / #4-03-1)
- **YL** (#50 / #4-03-1)
- **DZL** (#50 / #4-03-1)
- **DXL** (#50 / #4-03-1)
- **DYL** (#50 / #4-03-1)
- **LO** (#156 / #4-04-1)
- **dLO** (#156 / #4-04-1)
- **L-OVR** (#156 / #4-04-1)
- **dL-OVR** (#156 / #4-04-1)
- **LI** (#156 / #4-04-1)
- **dLI** (#156 / #4-04-1)
- **ALPHA** (#156 / #4-04-1)
- **B** (#156 / #4-04-1)

Further information: "Tool carrier reference point", Page 199

You can 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 321

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 558

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 inclination:

- M function **M128** (#9 / #4-01-1)

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 576

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

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393
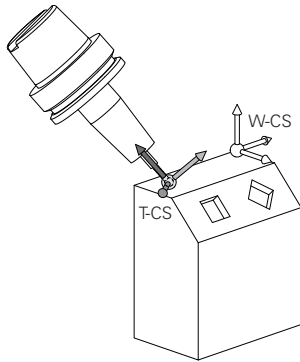
Use the miscellaneous function **M128** to define the tool inclination in the machine coordinate system **M-CS** using axis angles. The effects of the tool inclination depend on the machine kinematics:

Further information: "Notes", Page 579

11 L X+10 Y+45 A+10 C+5 R0 M128	; Straight line with miscellaneous function **M128** and axis angles

You can also define a tool inclination in the working plane coordinate system **WPL-CS** using spatial angles (e.g., with the **FUNCTION TCPM** function or a straight line **LN**).

11 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; **FUNCTION TCPM** with spatial angles
12 L A+0 B+45 C+0 R0 F2500	

11 LN X+48 Y+102 Z-1.5 **NX-0.04658107 NY0.00045007** **NZ0.8848844 TX-0.08076201** **TY-0.34090025 TZ0.93600126 R0** **M128**	; Straight line **LN** with surface normal vector and tool orientation

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management
  Further information: "Tool compensation for tool length and tool radius", Page 408
- Compensation values from the tool call
  Further information: "Tool compensation for tool length and tool radius", Page 408
- Values of the compensation tables ***.tco**
  Further information: "Tool compensation with compensation tables", Page 418
- Values of **FUNCTION TURNDATA CORR T-CS** (#50 / #4-03-1)
  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422
- 3D tool compensation with surface normal vectors (#9 / #4-01-1)
  Further information: "3D tool compensation (#9 / #4-01-1)", Page 424
- 3D tool radius compensation depending on the contact angle with compensation tables (#92 / #2-02-1)
  Further information: "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439

Position display (#44 / #1-06-1)

The display of the virtual tool axis **VT** refers to the tool coordinate system **T-CS**.

The control shows the values of **VT** in the **GPS** (#44 / #1-06-1) workspace and on the **GPS** tab of the **Status** workspace.

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

The HR 520 and HR 550 FS handwheels show the values of **VT** in the display.

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

11.2   NC functions for preset management

11.2.1   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

11.2.2   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.

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 management (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
	Number, text, or variable
	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 804

■ With the optional machine parameter **CfgColumnDescription** (no. 105607), the machine manufacturer specifies whether you can define the same content multiple times in the **DOC** column of the preset table. In this case, if you activate the preset using the **DOC** column, the control cannot clearly identify the desired preset. The control will display the error message **Table access failed**.

11.2.3 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 copy.

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 management (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
	Number, text, or variable
	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
	Number, text, or variable
	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!

With the optional machine parameter **CfgColumnDescription** (no. 105607), the machine manufacturer specifies whether you can define the same content multiple times in the **DOC** column of the preset table. This means that if you copy a preset using the **DOC** column, the control cannot clearly identify that preset. The control copies the preset with the lowest row number. So if another preset is copied instead of the desired one, there is a risk of collision in subsequent machining operations.

▶ Uniquely define the content of the **DOC** column

▶ Only copy the preset by indicating its row number

11.2.4 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

The control offsets the entered value against the actual table value. You can enter either a positive or a negative value.

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 input coordinate system **I-CS**. When correcting the OFFS values, the values reference the machine coordinate system **M-CS**.

Further information: "Reference systems", Page 310

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 management (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

Note

If you use **PRESET CORR** to correct the offset of a modulo axis, the control always writes a value in the module range −360° to +360° in the preset table.

If a rotary axis already has an offset outside of the modulo range, you can reduce the value with **PRESET CORR** and the entry **0** in the modulo range.

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.

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°.

11.3 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 832

■ 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**.

11.3.1 Activating a datum table in the NC program

To activate a datum table in the NC program:

Insert NC function	▶ Select **Insert NC function**
	> The control opens the **Insert NC function window.**
⊕	▶ Select **SEL TABLE**
	> The control opens the action bar.
📄	▶ Select **Selection**
	> A file selection window opens.
	▶ Select datum table
Select	▶ Select **Select**

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

Further information: "Settings in the Program workspace", Page 139

> ℹ 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

11.4 NC functions for coordinate transformation

11.4.1 Overview

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

Syntax	Meaning	Further information
TRANS DATUM	Shift the workpiece datum	Page 334
TRANS MIRROR	Mirror an axis	Page 336
TRANS ROTATION	Rotation about the tool axis	Page 338
TRANS SCALE	Scale contours and positions	Page 340
TRANS RESET	Reset the coordinate transformation	Page 341

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 321
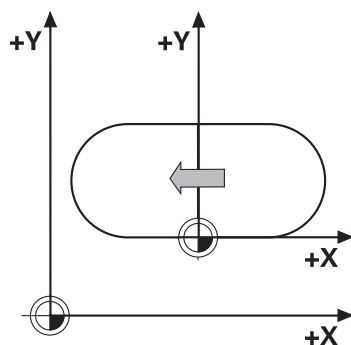
Related topics

- 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 348

- Reference systems

    Further information: "Reference systems", Page 310

11.4.2 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 832
* Activating the datum table

  Further information: "Activating a datum table in the NC program", Page 332
* Machine presets

  Further information: "Presets in the machine", Page 126

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 a datum table in the NC program", Page 332

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 ▶ Coordinate transformations TRANS ▶ TRANS DATUM

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS DATUM	Syntax initiator 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 **Parameter**	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 126

- 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 310

11.4.3 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
- Additive mirroring within the Global Program Settings GPS (#44 / #1-06-1)
  Further information: User's Manual for Setup and Program Run

Description of function

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

The control mirrors contours or positions about the active workpiece datum. If the datum is outside the contour, the control will also mirror the distance to the datum.

Further information: "Presets in the machine", Page 126
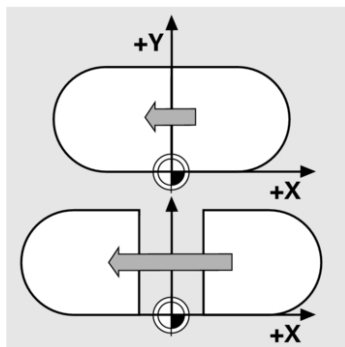
If you mirror only one axis, the machining direction of the tool is reversed. The rotational direction defined in a cycle will remain unchanged (e.g., if defined within one of the OCM cycles (#167 / #1-02-1)).
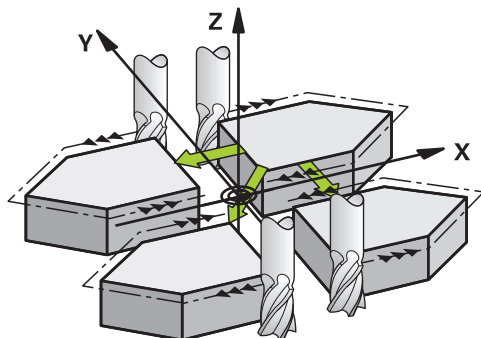
Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **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 124

You can select up to three axis values.
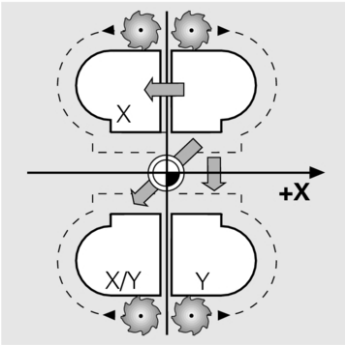
If mirroring is active, the control displays it on the **TRANS** tab of the **Status** workspace.

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

Input

| 11 TRANS MIRROR AXIS X | ; Mirror X coordinates about the Y axis |

To navigate to this function:

Insert NC function ▶ **All functions** ▶ **Special functions** ▶ **Functions** ▶ **Coordinate transformations TRANS** ▶ **TRANS MIRROR**

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS MIRROR	Syntax initiator 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 154
- 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 316

11.4.4 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
- Additive rotation within the Global Program SettingsGPS (#44 / #1-06-1)

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

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 126

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 124
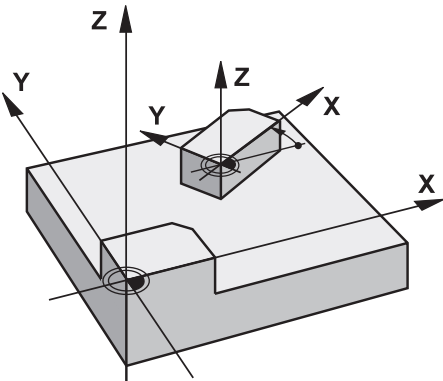
A rotation can be programmed as follows:

■ 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°

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Coordinate transformations TRANS ▶ TRANS ROTATION

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS ROTATION	Syntax initiator for a rotation
ROT or **RESET**	Enter an absolute or incremental angle of rotation or reset rotation Number or numerical parameter

Notes

- This function can be used only in the **FUNCTION MODE MILL** machining mode.

  Further information: "Switching the operating mode with FUNCTION MODE", Page 154

- 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

11.4.5 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 126
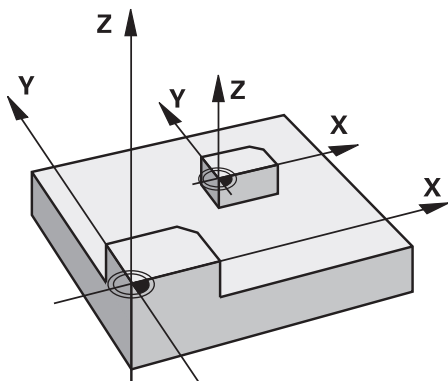
If you enter a scaling factor **SCL** less than 1, the contour will be reduced in size. If you enter a scaling factor **SCL** greater than 1, the contour will be enlarged.

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 |

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Coordinate transformations TRANS ▶ TRANS DATUM

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS SCALE	Syntax initiator for scaling
SCL or **RESET**	Enter the scaling factor or reset scaling
	Number or numerical parameter

Notes

- This function can be used only in the **FUNCTION MODE MILL** machining mode.

  Further information: "Switching the operating mode with FUNCTION MODE", Page 154

- 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.

11.4.6 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 functions for coordinate transformation", Page 333

- 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 334
Mirroring	**TRANS MIRROR**	Page 336
	Cycle **8 MIRRORING**	See the User's Manual for Machining Cycles
Rotation	**TRANS ROTATION**	Page 338
	Cycle **10 ROTATION**	See the User's Manual for Machining Cycles
Scaling	**TRANS SCALE**	Page 340
	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 ▶ Coordinate transformations TRANS ▶ TRANS RESET

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS RESET	Syntax initiator for resetting simple coordinate transformations

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

11.5.1 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 318
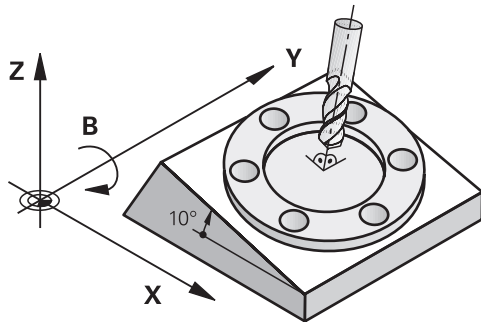
Two functions are available for tilting the working plane:

- 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 348

> **i** You can still execute 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 315

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 318
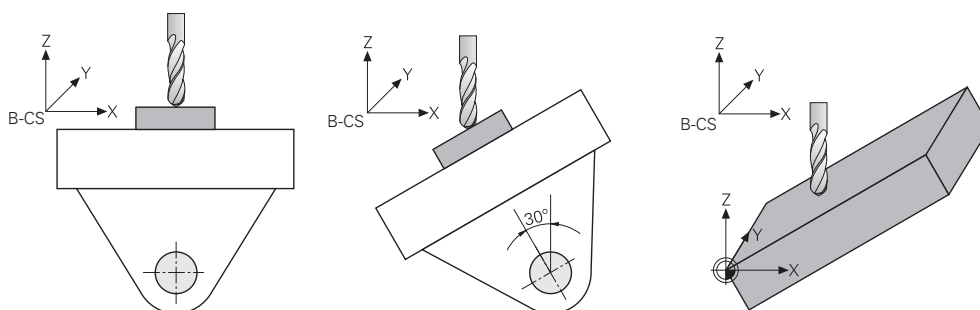
■ Machine with head rotary axes

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 318
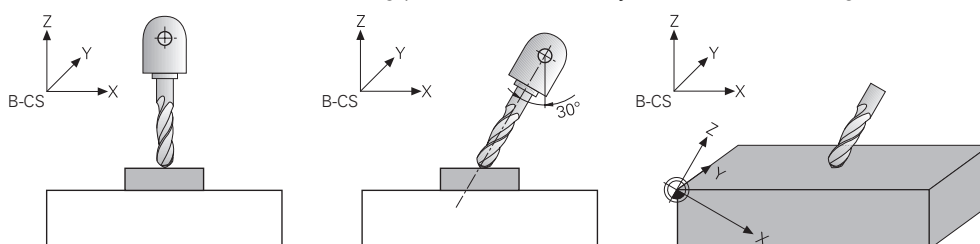
Difference between spatial angles and axis angles

Spatial angles

Using spatial angles, you can define the angle of the tool relative to the workpiece. While programming, there is no need to distinguish between head and table axes; often, the angles can be taken over directly from the drawing.

> **i** When you program using spatial angles, the machine kinematics need not be taken into account. This means that you can program as if only the tool was moving, just like in the **Workpiece** mode in the simulation.

The control takes care of calculating the required axis positions. This means that NC programs with spatial angles can also be used for other machines that might have other rotary axes.

The control might approach a defined spatial angle using different axis angles (e.g., +90° or −270°). The solution selected by the control may vary, depending on the machine. You can predefine a solution (e.g., by pre-positioning or by defining **SYM**).

When tilting with spatial angles, the control considers an active basic rotation or 3D basic rotation.

The control performs calculations with spatial angles in the following NC functions:

NC function	Further information
All **PLANE** functions except **PLANE AXIAL**	Page 348
Straight line **LN** with vectors (#9 / #4-01-1)	Page 425
FUNCTION TCPM with **AXIS SPAT** (#9 / #4-01-1) selected	Page 393

> **i** HEIDENHAIN recommends using spatial angles because of their greater flexibility.

Axis angles

Using axis angles, you can define an unambiguous position for a rotary axis. You can configure only the axes that are actually present on the machine. When programming with axis angles, you always need to take into account whether the rotary axis is arranged in the head or in the table. The programmed positions must be located in the traverse range of the machine.

If you program axis angles, the control cannot account for the basic rotation or 3D basic rotation in its calculation. Use offsets to orient the workpiece.

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

NC programs with axis angles can only be used for other machines that have the same rotary axes and matching traverse ranges.

Program axis angles in the following NC functions:

NC function	Further information
PLANE AXIAL	Page 378
M function **M128** (#9 / #4-01-1)	Page 576
FUNCTION TCPM with **AXIS POS** (#9 / #4-01-1) selected	Page 393

> **i** Even if **M128** or **FUNCTION TCPM** is active with **AXIS POS**, the control will use spatial angles (#9 / #4-01-1) anyway when calculating straight lines **LN** with vectors.

Example: Spatial angles vs. axis angles

The following example illustrates the difference between spatial angles and axis angles for tilted machining.

For machining, a machine with the table rotary axes **B** and **C** is used. The B axis is not arranged perpendicularly, but at a 45° angle in the left rear corner of the machine.

To machine a 45° chamfer at the front edge of the workpiece, program a spatial angle with **PLANE SPATIAL**.
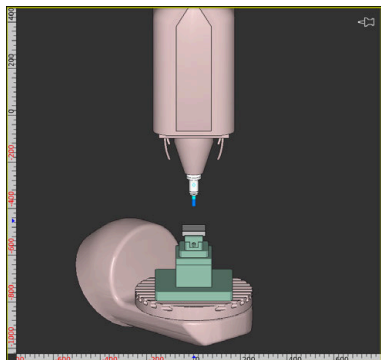
Machine kinematics (not tilted)   Workpiece with 45° chamfer
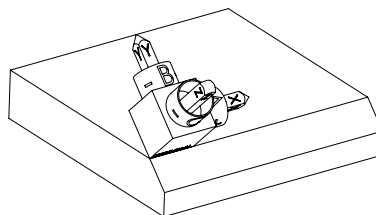
To mill a chamfer at the front edge, define the spatial angle **SPA+45**. The control calculates the required axis positions and rotates the B and C axes.

11 **PLANE SPATIAL SPA+45 SPB+0 SPC+0 MOVE FMAX**	; Tilt the working plane using a spatial angle

Position of rotary axes for **SPA+45**

In the **Positions** workspace, the control shows the positions for the **B** and **C** axes. If you program using axis angles, make sure to calculate and enter these axis positions.

11.5.2 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 541

- Adopting a tilted working plane in the **Manual** operating mode is possible 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 Adv. Function 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 310

> **i** 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 334

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 350

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 381

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 384

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 388

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

> **i** When deactivating or resetting the tilting function correctly, the icon indicating the tilted working plane must disappear.
>
> **Further information:** "PLANE RESET", Page 377

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 353
PROJECTED	Defines the working plane by means of two projection angles and one rotation angle	Page 358
EULER	Defines the working plane by means of three Euler angles	Page 362
VECTOR	Defines the working plane by means of two vectors	Page 365
POINTS	Defines the working plane by means of the coordinates of three points	Page 369
RELATIV	Defines the working plane by means of a single spatial angle with incremental effect	Page 373
AXIAL	Defines the working plane by means of a maximum of three absolute or incremental axis angles	Page 378
RESET	Resets tilting of the working plane	Page 377

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 **Single Block** 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 positioning. There is a danger of collision while the axis moves out of the coupling and during the positioning 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.

> ℹ️ 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 373
- Entering the axis angle
  Further information: "PLANE AXIAL", Page 378

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.
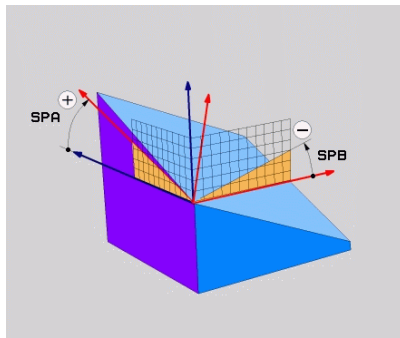
Spatial angles **SPA** and **SPB**
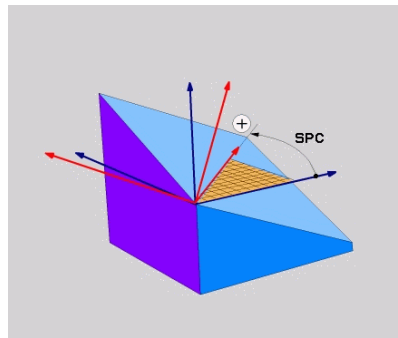
Spatial angle **SPC**

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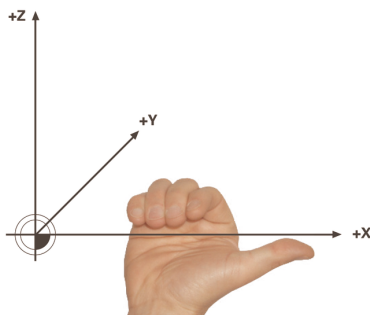
The thumb of your right hand points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

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 356

> **i** 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 373

Application example

Example

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
```
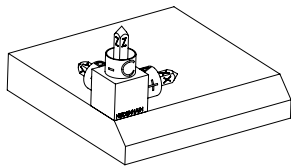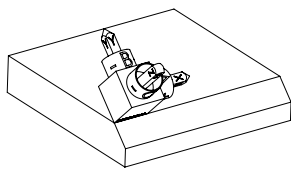
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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.

> **i** When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following spatial angles:
> - **SPA+45**, **SPB+0** and **SPC+90** for the second chamfer
>   **Further information:** "Notes", Page 356
> - **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	Syntax initiator for defining 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
	ℹ️ Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined.
	Further information: "Rotary axis positioning", Page 381
SYM or **SEQ**	Select an unambiguous tilting solution
	Further information: "Tilting solution", Page 384
	Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type
	Further information: "Transformation types", Page 388
	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
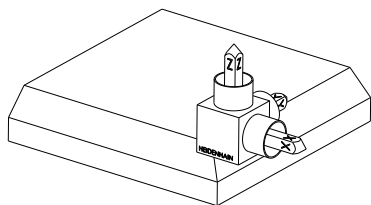
Initial state
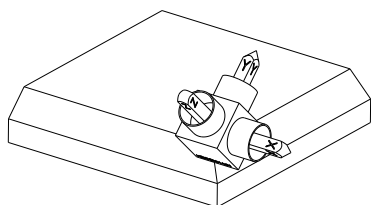
SPA+45

Orientation of tool axis **Z**

Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**
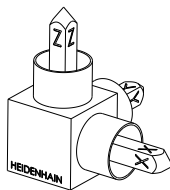
SPB+0

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

No rotation with value 0
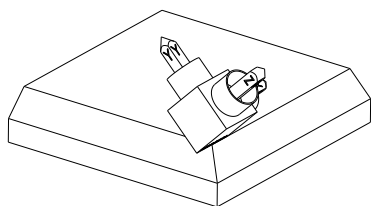
SPC+90

Orientation of main axis **X**

Rotation around the Z axis of the non-tilted **W-CS**
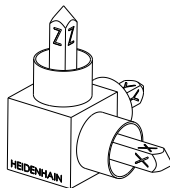
View C-B-A
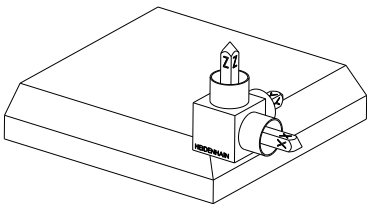
Initial state
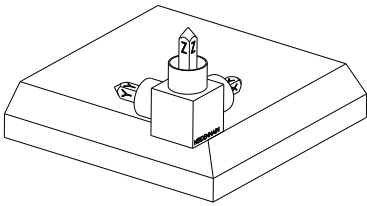
SPC+90

Orientation of main axis **X**

Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane
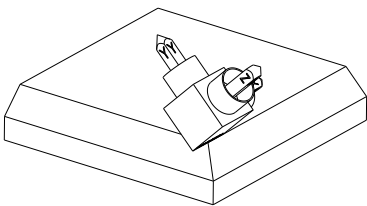
SPB+0

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 124

Use an additional rotation angle to optionally align the X axis in the tilted working plane.
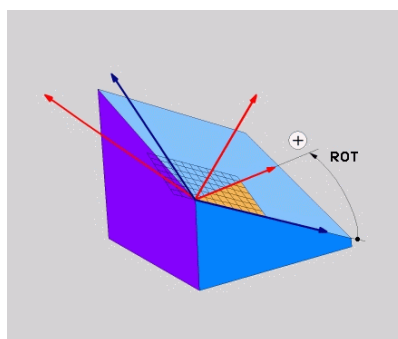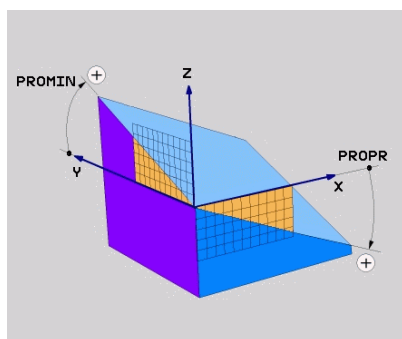
Projection angles **PROMIN** and **PROPR**

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.

> ℹ️ 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 361

Application example

Example

```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE
   ROT
```
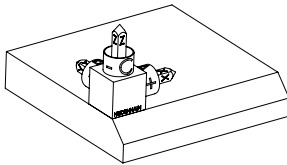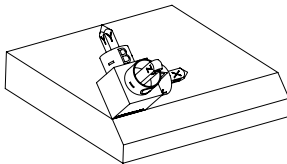
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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 (i) Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 381
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 384 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 388 Optional syntax element

Notes

Procedure in case of hidden workpiece edges, using the example of a diagonal hole
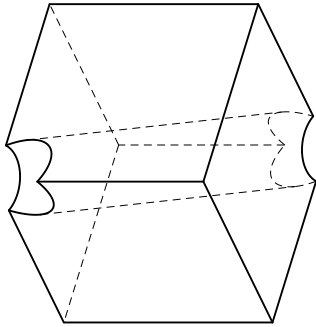
Cube with a diagonal hole
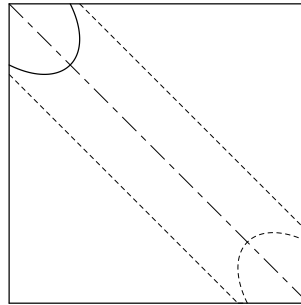
Front view, meaning projection on the **ZX** working plane

Example

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

Comparison of projection and spatial angles
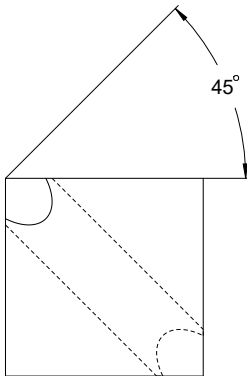
When imagining the workpiece to be transparent, the projection angles are easy to find.

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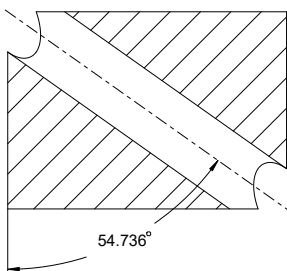
When defining the working plane by using spatial angles, the spatial diagonal must be considered.

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, for example, 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.
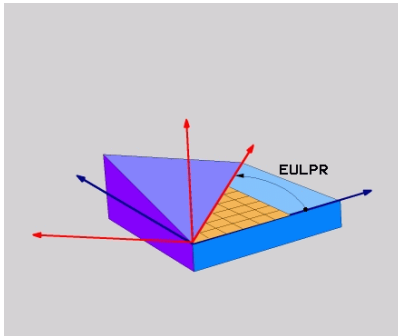
Euler angle **EULPR**
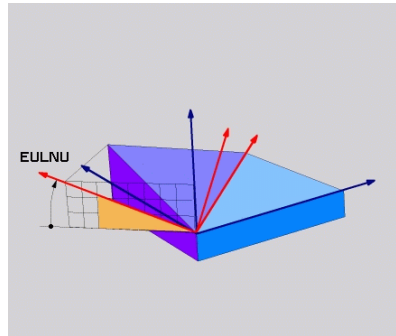
Euler angle **EULNU**
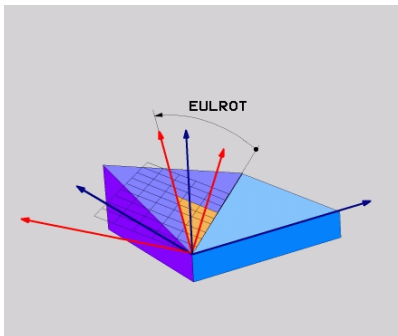
Euler angle **EULROT**

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.

> ℹ 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 373
>
> 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 353
>
> **Further information:** "Rotations with TRANS ROTATION", Page 338

Application example

Example

11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
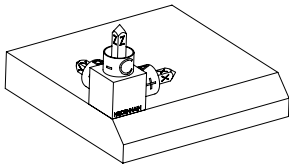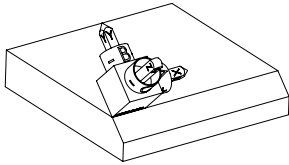
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which 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 ℹ Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 381
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 384 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 388 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 539

Description of function

Vectors define a working plane as two independent specifications of direction, starting from the non-tilted workpiece coordinate system **W-CS**.
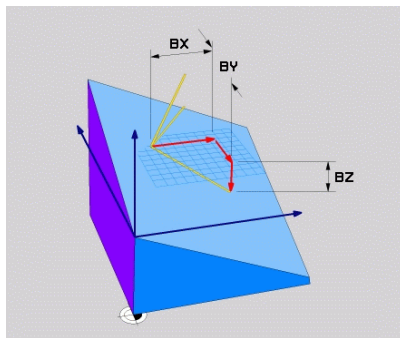
Base vector with components **BX**, **BY** and **BZ**
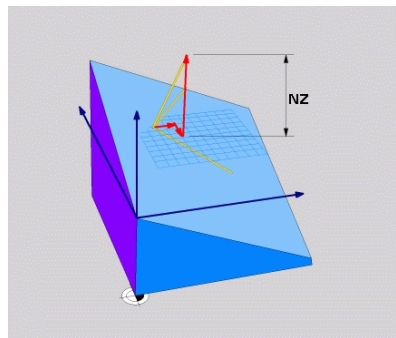
NZ component of the normalized vector

All six components must be defined even if one or several components equals 0.

> ℹ️ 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 366

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**
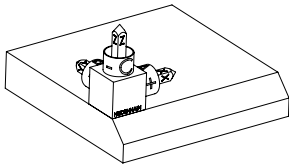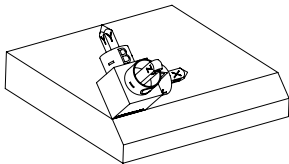
Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around 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 ℹ️ Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 381
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 384 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 388 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: Measuring Cycles for Workpieces and Tools User's Manual

Description of function

Points define a working plane by using their coordinates in the non-tilted workpiece coordinate system **W-CS**.
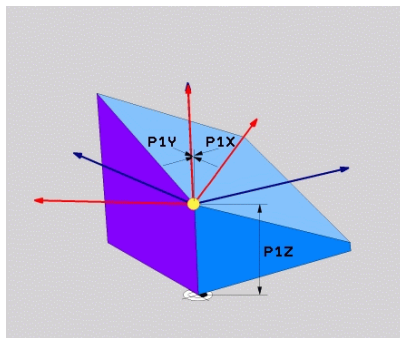
First point with coordinates **P1X**, **P1Y** and **P1Z**
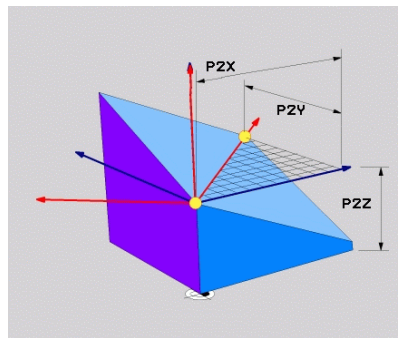
Second point with coordinates **P2X**, **P2Y** and **P2Z**
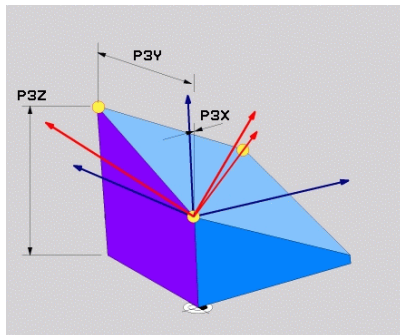
Third point with coordinates **P3X**, **P3Y** and **P3Z**

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

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

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

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

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

The third point with coordinates **P3X**, **P3Y** and **P3Z** defines the slope of the tilted working plane.
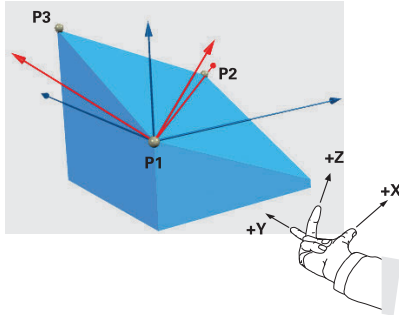
To direct the positive tool axis direction away from the workpiece, the following conditions apply to the position of the three points:

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

Application example

Example

```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```
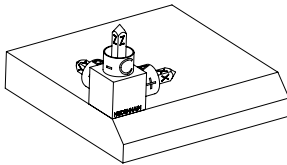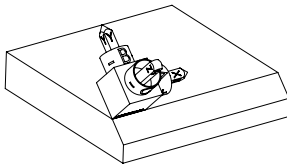
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis	Using the first two points **P1** and **P2**, the control orients the X axis of the **WPL-CS**. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. **P3** defines the slope of the tilted working plane. The orientations of the tilted Y and Z axes result automatically because all axes are perpendicular to one another.

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

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

Input

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

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE POINTS	Syntax initiator for the working plane definition by means of three points
P1X, **P1Y** and **P1Z**	Coordinates of the first point of the tilted X axis, referenced to the workpiece coordinate system **W-CS** Input: **-999999999.999999...+999999999.999999**
P2X, **P2Y** and **P2Z**	Coordinates of the second point, referenced to the **W-CS** for orienting the tilted X axis Input: **-999999999.999999...+999999999.999999**
P3X, **P3Y** and **P3Z**	Coordinates of the third point, referenced to the **W-CS** for inclining the tilted working plane Input: **-999999999.999999...+999999999.999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning ⓘ Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 381
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 384 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 388 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 310

Description of function

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

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

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

> ℹ️ **PLANE RELATIV** allows, for example, programming a chamfer on a tilted workpiece surface by tilting the working plane further by the chamfer angle.
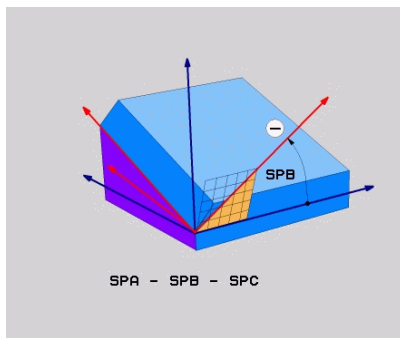
Additive spatial angle **SPB**

Each **PLANE RELATIV** function defines one spatial angle exclusively. However, it is possible to program any number of **PLANE RELATIV** functions in a row.

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

Application example

Example

11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
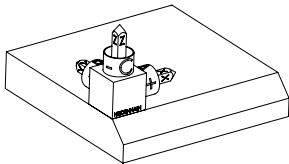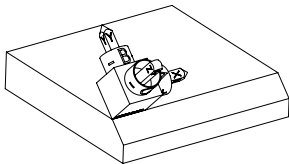
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis	Using the spatial angle **SPA+45**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **SPA** angle is around the non-tilted X axis.
	The orientation of the tilted X axis equals the orientation of the non-tilted X axis.
	The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

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

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

i When shifting the workpiece datum further in a tilted working plane, incremental values must be defined.

Further information: "Note", Page 376

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 381
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 384 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 388 Optional syntax element

Note

Incremental datum shift using a chamfer as example
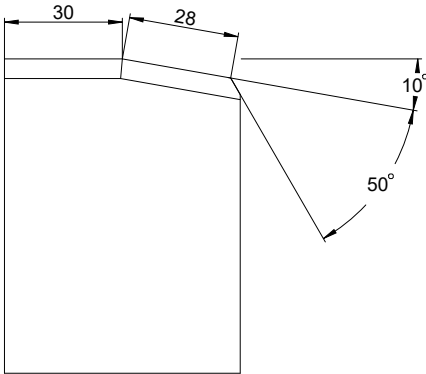
50° chamfer on a tilted workpiece
surface

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

> **i** 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 381

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

> **i** Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined.

Further information: "Rotary axis positioning", Page 381

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

> **i** The status display allows checking the desired status of the tilting situation.
> **Further information:** "Status display", Page 349

- The touch probe functions allow you to save the inclined position of the workpiece to the preset table as a 3D basic rotation such as **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 the machining, since the control does not consider the 3D basic rotation for this function.
  Further information: "PLANE SPATIAL", Page 353

PLANE AXIAL

Application

Use the **PLANE AXIAL** function to define the working plane with anywhere from one to three absolute or incremental axis angles.

An axis angle can be programmed for each rotary axis available on the machine.

> **i** Because you are able to define just one axis angle, you can also use **PLANE AXIAL** on machines with just one rotary axis.

Please note that NC programs with axis angles always depend on the kinematics and therefore depend on the machine in question!

Related topics

- Programming independently of kinematics, using spatial angles

  Further information: "PLANE SPATIAL", Page 353

Description of function

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

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

As the axis angles depend on the kinematics, a distinction must be made between the head and the table axes as far as the algebraic signs are concerned.
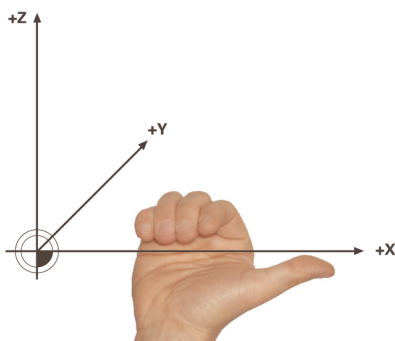
Extended right-hand rule for head rotary axes
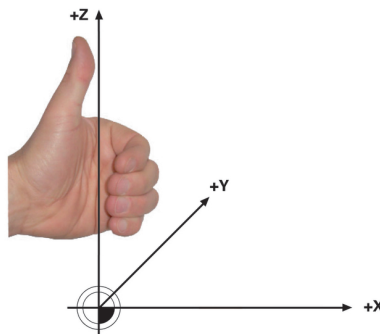
Extended left-hand rule for table rotary axes

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

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

Application example

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

Example

11 PLANE AXIAL A+45 TURN MB MAX FMAX
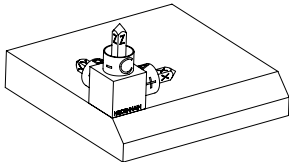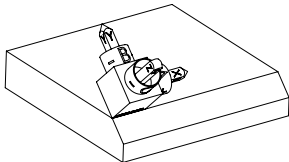
Initial state	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis	Using the defined axis angle **A**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by angle **A** is around the non-tilted X axis.

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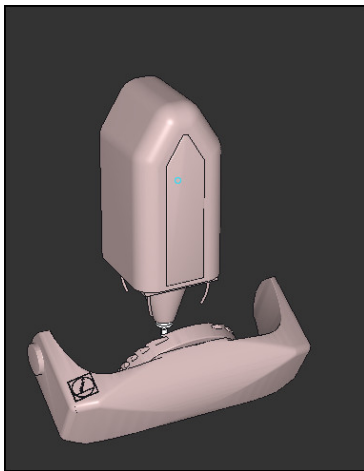
The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

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

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

Input

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

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE AXIAL	Syntax initiator for the working plane definition using one to three axis angles
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 381

> ⓘ 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 316

- 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.
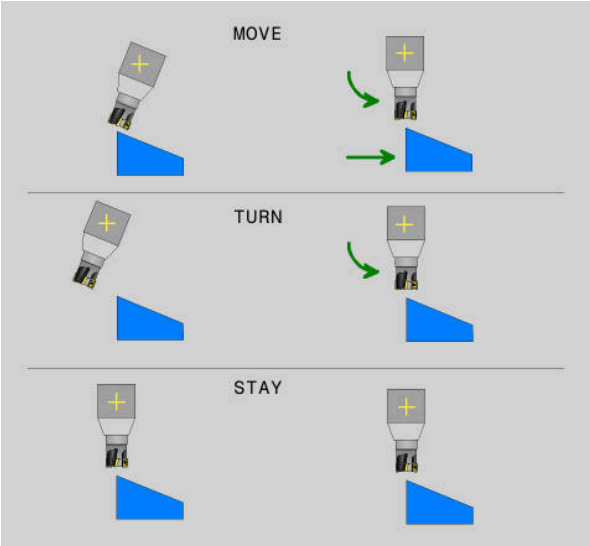
Type of rotary axis positioning	Meaning
MOVE	If you perform tilting near the workpiece, then use this option. **Further information:** "Rotary axis positioning with MOVE", Page 382
TURN	If the workpiece is so large that the range of traverse is not sufficient for the compensating movement of the linear axes, then use this option. **Further information:** "Rotary axis positioning TURN", Page 382
STAY	The control does not position any axes. **Further information:** "Rotary axis positioning with STAY", Page 383

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.

NOTICE

Danger of collision!

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

▶ Ensure sufficient distance between the tool and the workpiece

---

When **DIST** is not defined or when you define the value 0, the center of rotation and consequently the center of the compensation movements is in the tool tip.

When you define **DIST** with a value greater than 0, the center of rotation in the tool axis is shifted away from the tool tip by this value.

ⓘ If you wish to tilt about a certain point on the workpiece, ensure the following:

- Prior to tilting to position, the tool is positioned directly above the desired point on the workpiece.
- The value defined in **DIST** matches exactly the clearance between the tool tip and the desired center of rotation.
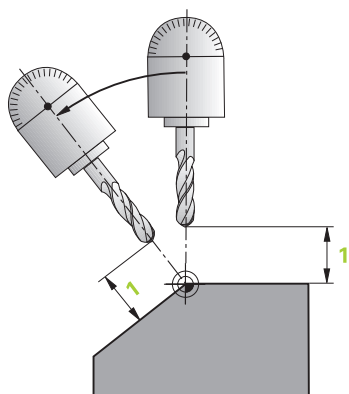
Rotary axis positioning TURN

The control positions only the rotary axes. The tool must be positioned after tilting to position.

Rotary axis positioning with STAY

Both the rotary axes and the tool must be positioned after tilting to position.

> **i** Even with **STAY**, the control orients the working plane coordinate system **WPL-CS** automatically.

When selecting **STAY**, the rotary axes must be tilted to position in a separate positioning block after the **PLANE** function.

In the positioning block, use only the axis angles calculated by the control:

- **Q120** for the axis angle of the A axis
- **Q121** for the axis angle of the B axis
- **Q122** for the axis angle of the C axis

The variable avoids entry and calculating errors. In addition, no changes are required after changing the values within the **PLANE** functions.

Example

```
11 L A+Q120 C+Q122 FMAX
```

Input

MOVE

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 MOVE DIST0 FMAX
```

Selecting **MOVE** allows defining the syntax elements below:

Syntax element	Meaning
DIST	Distance between center of rotation and the tool tip
	Input: **0...99999999.9999999**
	Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning
	Optional syntax element

TURN

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX
```

Selecting **TURN** allows defining the syntax elements below:

Syntax element	Meaning
MB	Retraction in the current tool axis direction before positioning the rotary axis
	Values with an incremental effect can be entered or a retraction up to the traverse limit can be defined by selecting **MAX**.
	Input: **0...99999999.9999999** or **MAX**
	Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning
	Optional syntax element

STAY

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 STAY
```

Selecting **STAY** does not allow defining further syntax elements.

Note

NOTICE
Danger of collision!
The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect or no pre-positioning before tilting the tool into position can lead to a risk of collision during the tilting movement!
▶ Program a safe position before the tilting movement
▶ Carefully test the NC program or program section in the **Single Block** mode

Tilting solution

Application

A defined spatial angle can be reached by performing various rotary axes movements.

In order to achieve a certain solution, for example to avoid collisions, you can pre-position the rotary axes or program **SYM (SEQ)**.

Further information: "Difference between spatial angles and axis angles", Page 345

Description of function

In the optional machine parameter **prohibitSEQ** (no. 201209), the machine manufacturer defines whether the control will provide the **SEQ** option in addition to **SYM**.

> **i** HEIDENHAIN recommends using **SYM** because this function is suitable for all machine kinematics.
>
> Please note that you must not simply replace **SEQ** with **SYM**, because the behavior is different depending on the machine kinematics.

Option	Meaning
SYM	With **SYM**, you select a tilting solution relative to the symmetry point of the primary rotary axis. The symmetry point lies at the center of the rotary axis. The primary rotary axis is the first rotary axis, seen from the tool, or the last rotary axis, seen from the table. **Further information:** "Tilting solution SYM", Page 386
SEQ	With **SEQ**, you select a tilting solution relative to the basic position of the primary rotary axis. Using **SEQ** only makes sense for machine kinematics where the symmetry point of the primary rotary axis lies at 0°. **Further information:** "Tilting solution SEQ", Page 386
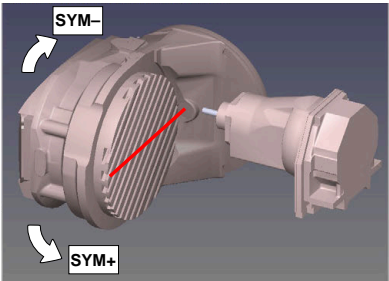
Reference point for **SYM**: A = −90°   Reference point for **SEQ**: A = 0°
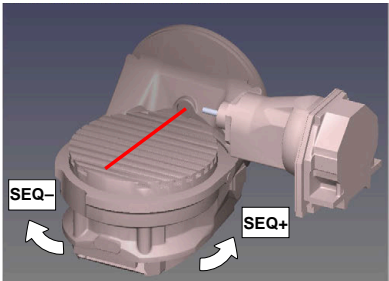
Entering **SYM** or **SEQ** is optional.

If you do not define **SYM** (**SEQ**), the control will select the solution variant with the smallest number of rotary axis movements, based on the current rotary axis positions.

If no tilting solution is available within the traverse range of the machine, the control will issue the **Entered angle not permitted** error message. This happens regardless of whether you have defined **SYM** (**SEQ**) or not.

Tilting solution SYM

Using the **SYM** function, you select a solution option relative to the symmetry point of the primary rotary axis:

- **SYM+** positions the axis in the positive half-space
- **SYM-** positions the axis in the negative half-space

As opposed to **SEQ**, **SYM** uses the symmetry point of the primary rotary axis as the reference point. Every rotary axis has two symmetry positions, which are spaced by 180°. In some cases, only one symmetry position is within the traverse range of the machine.

> **ℹ** Determine the symmetry point as follows:
> - ▶ Perform **PLANE SPATIAL** with any spatial angle and **SYM+**
> - ▶ Save the axis angle of the primary rotary 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

Using the **SEQ** function, you select one of the solution options relative to the basic position of the primary rotary axis:

- **SEQ+** positions the axis in the positive tilting area
- **SEQ-** positions the axis in the negative tilting area

SEQ is based on the basic position 0° of the primary rotary axis. If both solution options are within the positive or negative area, the control will use the tilting solution that requires the smallest number of rotary axis movements, relative to the current position. If you need the second-best solution, then either pre-position the rotary axes (in the area of the second solution) before tilting the working plane, or use **SYM**.

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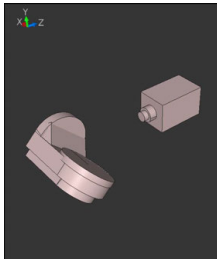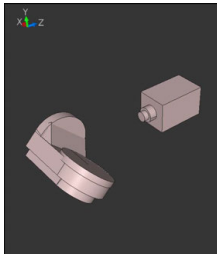
Machine with C rotary axis and A tilting table.
Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

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 changing 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.
>
> The transformation types work for all tilting functions except **PLANE AXIAL**.

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	For example, **TABLE ROT** works with **PLANE SPATIAL** as follows:
	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
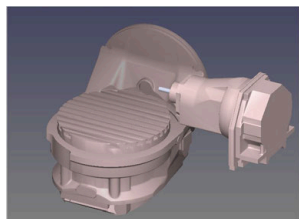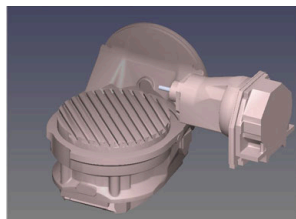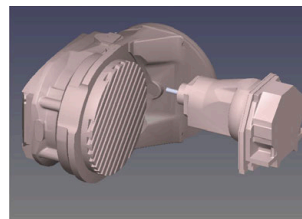
Origin   A = 0, B = 45   A = −90, B = 45

> 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**).

11.6 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 393
- 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 576
- Tilting the working plane (#8 / #1-01-1)
  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 343
- Presets on the tool
  Further information: "Presets on the tool", Page 199
- Reference systems
  Further information: "Reference systems", Page 310

Requirements

- Machine with rotary axes
- Kinematics description
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Adv. Function Set 2 (#9 / #4-01-1) software option

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 343

Inclined machining can be implemented using the following functions:

- Incremental traverse of rotary axis

  Further information: "Inclined machining with incremental process", Page 392

- Normal vectors

  Further information: "Inclined machining using normal vectors", Page 392

Inclined machining with incremental process

Inclined machining can be programmed by changing the inclination angle in addition to normal linear movement while function **FUNCTION TCPM** or **M128** is active (e.g., **L 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
* - ...	

11.7 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. While **FUNCTION TCPM** is active, the control compensates for changed tool inclinations by performing compensation movements of the linear axes. This means that you can change the tool inclination during machining without damaging the contour.

> **i** **FUNCTION TCPM** is an improvement of miscellaneous function **M128**.
> 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 576
- Tilting the working plane
  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 343
- Presets on the tool
  Further information: "Presets on the tool", Page 199
- Reference systems
  Further information: "Reference systems", Page 310

Requirements

- Machine with rotary axes
  Depending on the mechanical design of the rotary axes, not all features might be available (e.g., no simultaneous machining). Refer to your machine manual.
- Control prepared by the machine manufacturer
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Adv. Function Set 2 software option (#9 / #4-01-1)

Description of function

Behavior without **TCPM**

Behavior with **TCPM** (**REFPNT CENTER-CENTER**)
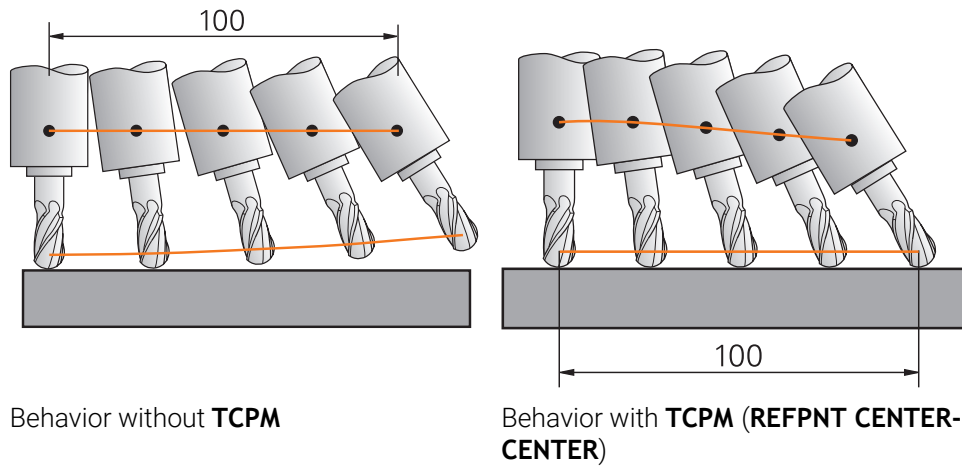
If **FUNCTION TCPM** is active, the control shows the **TCPM** icon in the position display.

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

While **FUNCTION TCPM** is active, the following NC functions cannot be used as usual or not at all:

- **M91**/**M92**
- **TOOL CALL**
- Tool radius compensation **RL**/**RR**

  If **FUNCTION TCPM** is active, this function will only define the direction for 3D radius compensation.

  For CAM-generated NC programs, program **FUNCTION PROG PATH IS CONTOUR** instead.

- Tool-tip radius compensation SRK for turning programs (#50 / #4-03-1)

  Only possible for **FUNCTION TCPM** with **REFPNT TIP-CENTER**

The **FUNCTION RESET TCPM** function resets the **FUNCTION TCPM** function.

Input

FUNCTION TCPM

> 10 FUNCTION TCPM F CONT AXIS SPAT PATHCTRL AXIS REFPNT CENTER-CENTER
>   F1000

To navigate to this function:

Insert NC function ▶ Special functions ▶ Functions ▶ Tool inclination compensation TCPM ▶ FUNCTION TCPM

The NC function includes 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 396
AXIS POS or **AXIS SPAT**	Interpretation of programmed rotary axis coordinates as axis angles or spatial angles **Further information:** "Rotary axis coordinates programmed as axis or spatial angles", Page 397
PATHCTRL AXIS or **PATHCTRL VECTOR**	Interpolation of tool angle of inclination **Further information:** "Interpolation of tool angle of inclination between starting and end points", Page 398
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 399 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 400 Optional syntax element

FUNCTION RESET TCPM

> 10 FUNCTION RESET TCPM

To navigate to this function:
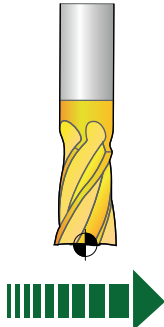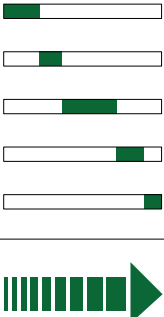
Insert NC function ▶ Special functions ▶ Functions ▶ Tool inclination compensation TCPM ▶ FUNCTION RESET TCPM

The NC function includes 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	Meaning
F TCP	The control interprets the programmed feed rate as the velocity value of the tool location point. The control calculates the required feed rate for the individual axes automatically and keeps the feed rate at the tool location point constant.
	If the ratio of linear and rotary axis movements in an NC block is balanced, **F TCP** will usually produce a better surface in face milling. If the NC block defines significantly more rotary axis movements than linear axis movements, the rotary axes need to be positioned very quickly. In order to keep the feed rate at the tool location point constant in this case, a dynamic machine is required.
F CONT	The control interprets the programmed feed rate as a vectorial axis feed rate. The programmed feed rate will be subdivided into components, taking all programmed axis movements in the NC block into account. The control calculates the velocity value of the compensation movement in the linear axes independent of the programmed feed rate.
	F CONT protects the machine because the axes will be accelerated more smoothly. This will generate feed-rate variations at the tool location point.
	Program **F CONT**, for example, if you need to change the tool inclination while the tool is not in contact with the workpiece.
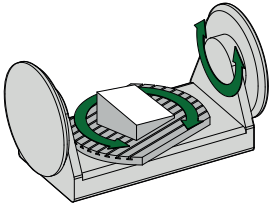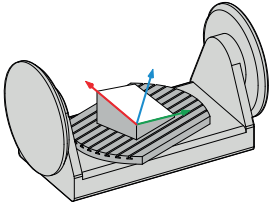
> **i** You can limit the velocity of the compensation movements in the linear axes with the **F** syntax element.
>
> **Further information:** "Limiting the linear-axis feed rate", Page 400

Rotary axis coordinates programmed as axis or spatial angles

The control can interpret the programmed rotary axis coordinates in the following ways:

Selection	Meaning
AXIS POS	The control interprets the programmed rotary axis coordinates as axis angles. The control positions the rotary axes to the positions defined in the NC program.
	NC programs with axis angles can only be used for other machines that have the same rotary axes and matching traverse ranges.
	You cannot program a basic rotation or 3D basic rotation with **AXIS POS**, and **FUNCTION TCPM** cannot be used if the working plane is tilted.
AXIS SPAT	The control interprets the programmed rotary axis coordinates as spatial angles.
	The control takes care of calculating the required axis positions. This means that NC programs with spatial angles can also be used for other machines that might have other rotary axes.
	With **AXIS SPAT**, you can orient the workpiece using a basic rotation or 3D basic rotation and use **FUNCTION TCPM** in case the working plane is tilted.

Further information: "Difference between spatial angles and axis angles", Page 345
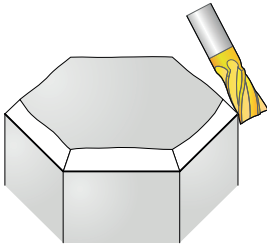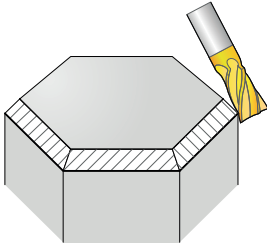
> ⓘ ■ The machine manufacturer defines in the kinematics description whether you can use **AXIS SPAT** to also program axes that do not exist physically on the machine. The control saves this information in the machine parameter **progAxes** (no. 202802).
>
> ■ A programmed tool inclination will not tilt the working plane, as with the **PLANE** functions, for example. This means that you can program **FUNCTION TCPM** with **AXIS SPAT** even if the working plane is tilted.
>
> ■ You can program **FUNCTION TCPM** with **AXIS POS** manually only for machines with perpendicular kinematics. With other machine kinematics, you need a CAM system to calculate the correct values (e.g., for 45° swivel heads).
>
> ■ **M128** and **FUNCTION TCPM** with **AXIS POS** selected do not take an active basic rotation or 3D basic rotation into account. Program **FUNCTION TCPM** with **AXIS SPAT** selected, or CAM outputs with **LN** straight lines and a tool vector.
>
> **Further information:** "Straight line LN", Page 425

Interpolation of tool angle of inclination between starting and end points

The control provides two ways to calculate the path of the rotary axes between the starting and end points.

In both cases, the tool location point will be moved directly and the tool will be positioned, with the programmed tool inclination, at the end point defined in the NC block.

Selection	Meaning
PATHCTRL AXIS 	The control calculates the rotary axis positions for the end point. During the movement, the control will position the rotary axes using a direct path. Depending on the program and kinematics, **PATHCTRL AXIS** might not produce a planar surface area when performing peripheral milling. **PATHCTRL AXIS** can be used, for example, for face milling with a spherical cutter.
PATHCTRL VECTOR 	The control calculates a plane using the tool inclination at the starting and end points and maintains the plane during traverse. If the direct traverse path deviates from the plane, the control will compensate for this deviation with additional rotary axis movements. You can use **PATHCTRL VECTOR** for peripheral milling in order to obtain a planar cylindrical surface even if the tool inclination is changed.
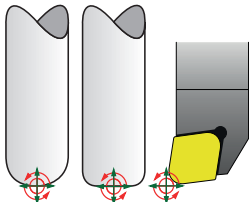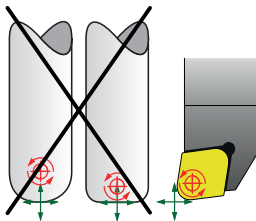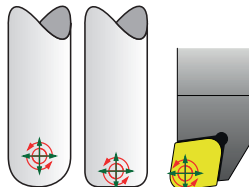
> ⓘ
> - If **PATHCTRL AXIS** is used, the axis movements are smoother and machining times might be shorter. **PATHCTRL VECTOR** should only be used if you cannot obtain the desired result with **PATHCTRL AXIS**.
> - When programming **PATHCTRL AXIS**, you can specify a **Tolerance for rotary axes TA** in Cycle **32 TOLERANCE** to obtain an even smoother movement.
> **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	Meaning
REFPNT TIP-TIP	The tool location point and the tool rotation point are at the tool tip. You can use **REFPNT TIP-TIP** with end mills, for example, for peripheral milling. **REFPNT TIP-TIP** is the default setting.
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 TIP-CENTER** has been optimized for turning tools (#50 / #4-03-1). When the control positions the rotary axes, the tool rotation point remains at the same position. This allows you to machine, for example, complex contours by simultaneous turning. **Further information:** "Theoretical tool tip TIP for tool radius compensation", Page 416
REFPNT CENTER-CENTER	The tool location point and the tool rotation point are located at the tool center point. **REFPNT CENTER-CENTER** can be used for face milling with spherical cutters. 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.
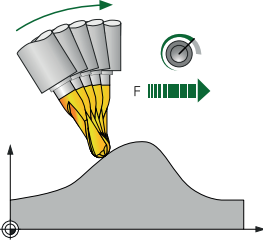
Further information: "Presets on the tool", Page 199

> ℹ️ ■ If you program **REFPNT CENTER-CENTER**, the control can monitor the entire tool length for collisions during machining.
> If you want to use **M128** in the same way as **REFPNT CENTER-CENTER**, you need to program the tool with **DL** in order to shorten tool radius 2 **R2**. In this case, the control will not monitor the remaining tool length for collisions.
> ■ 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

Limiting the linear-axis feed rate

Graphs	Meaning
	The optional input of **F** allows you to limit the feed rate of compensation movements of the linear axes. The feed rate of the programmed linear motions does not change.
	Thus, you can avoid fast compensation movements (e.g., in case of retraction movements at rapid traverse).
	The linear axis feed-rate limit remains in effect until you program a new value or reset **FUNCTION TCPM**.

> **i** Make sure to select a value for the linear axis feed-rate limit that is not too small because large feed-rate variations may occur at the tool location point. Feed-rate variations impair the surface quality.
>
> If **FUNCTION TCPM** is active, the feed-rate limit will only be effective for movements with a rotary-axis component, not for entirely linear motions.

Notes

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable positioning. There is a danger of collision while the axis moves out of the coupling and during the positioning operation!

▶ Make sure to retract the tool before changing the position of the rotary axis

---

- If you always select the first selection option offered for **FUNCTION TCPM**, you will achieve the same functionality as with **M128**. In this case program the syntax **FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-TIP**.

- 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 579

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. For **FUNCTION TCPM** and **M128** the machine parameter applies only to one rotary axis of the table 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 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 316

- 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.

11.7.1 Program structure with FUNCTION TCPM

Here you see a possible program structure with **FUNCTION TCPM**. You can use this structure for various machining operations.

	BLK FORM…	
	TOOL CALL…	
Shift the datum, if required	**TRANS DATUM…**	; e.g., for using it as a datum for a tilted working plane
Tilt the working plane, if required	**PLANE SPATIAL…**	; Only possible for **FUNCTION TCPM** with **AXIS SPAT**
Pre-positioning	**L X… Y… Z…**	
Activate FUNCTION TCPM	**FUNCTION TCPM…**	
Define the tool inclination	**L A…**	
Machine the contour with TCPM	**L X…**	
	LN…	
	L A…	; Reset the tool inclination
Deactivate FUNCTION TCPM	**FUNCTION RESET TCPM**	; Alternatively **M129**
Reset the datum shift	**TRANS RESET**	
Reset the tilted working plane	**PLANE RESET…**	
…		

11.7.2 Example: Machining a chamfer with FUNCTION TCPM
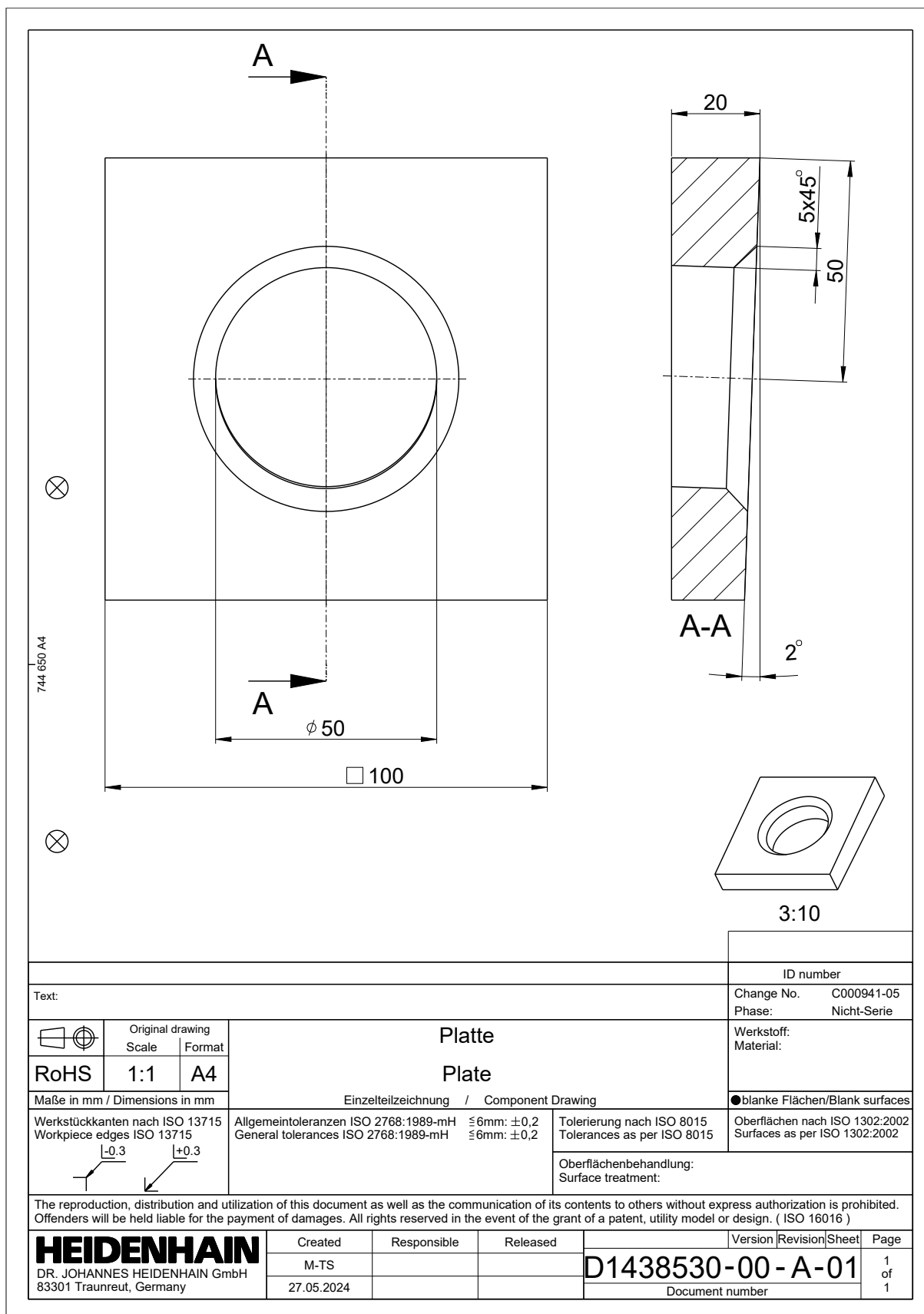
This NC program is structured as shown above.

0 BEGIN PGM 1438530 MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-20	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 ;	
4 * -	; Main program
5 TOOL CALL "MILL_D20_ROUGH" Z S5000 F1000	
6 CALL PGM TNC:\nc_prog\SAFE.h	
7 M3	
8 CALL LBL "RESET"	
9 CALL LBL "PLANE"	
10 ;	
11 CYCL DEF 233 FACE MILLING ~	
Q215=+1 ;MACHINING OPERATION ~	
Q389=+2 ;MILLING STRATEGY ~	
Q350=+2 ;MILLING DIRECTION ~	
Q218=+100 ;FIRST SIDE LENGTH ~	
Q219=+110 ;2ND SIDE LENGTH ~	
Q227=+10 ;STARTNG PNT 3RD AXIS ~	
Q386=+0 ;END POINT 3RD AXIS ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q202=+10 ;MAX. PLUNGING DEPTH ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q207=AUTO ;FEED RATE MILLING ~	
Q385=AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q357=+2 ;CLEARANCE TO SIDE ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q347=+0 ;1ST LIMIT ~	
Q348=+0 ;2ND LIMIT ~	
Q349=+0 ;3RD LIMIT ~	
Q220=+0 ;CORNER RADIUS ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q367=+4 ;SURFACE POSITION	
12 L X-50 Y+0 Z+5 R0 FMAX M99	
13 ;	
14 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+0 ;MACHINING OPERATION ~	
Q223=+50 ;CIRCLE DIAMETER ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q207=AUTO ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	

Q201=-20	;DEPTH ~	
Q202=+20	;PLUNGING DEPTH ~	
Q369=+0	;ALLOWANCE FOR FLOOR ~	
Q206=AUTO	;FEED RATE FOR PLNGNG ~	
Q338=+0	;INFEED FOR FINISHING ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+50	;2ND SET-UP CLEARANCE ~	
Q370=+1	;TOOL PATH OVERLAP ~	
Q366=+1	;PLUNGE ~	
Q385=+500	;FINISHING FEED RATE ~	
Q439=+0	;FEED RATE REFERENCE	
15 L X+0 Y-50 R0 FMAX M99		
16 CALL LBL "RESET"		
17 ;		
18 TOOL CALL "MILL_D12_ROUGH" Z S5000 F1000		
19 CALL PGM TNC:\nc_prog\SAFE.h		
20 M3		
21 CALL LBL "PLANE"		
22 ;		
23 * -	; Simultaneous milling of the chamfer	
24 TRANS DATUM AXIS IX+25 IY-50 IZ-5	; Shift the datum to the lower chamfer edge	
25 L X-20 Y+0 Z-1 R0 FMAX	; Pre-position	
26 PLANE RELATIV SPB+45 MOVE	; Tilt the working plane for pre-positioning	
27 L X-Q108	; Move to machining position	
28 PLANE RELATIV SPB-45 STAY	; Reset tilting mathematically	
29 FUNCTION TCPM F CONT AXIS SPAT PATHCTRL AXIS REFPNT TIP-TIP	; Activate **FUNCTION TCPM**	
30 L B+45	; Pre-position the tool	
31 TRANS DATUM AXIS X+0 IZ+5	; Shift the datum to the center of the circular pocket	
32 CC X+0 Y+0		
33 CP IPA-90 C-90 DR- F AUTO	; Machine the chamfer	
34 CP IPA-90 IC-90 DR-		
35 CP IPA-90 IC-90 DR-		
36 CP IPA-90 IC-90 DR-		
37 DEP LCT X+0 Y+0 R3	; Depart from the contour	
38 L B+0	; Reset the tool inclination	
39 ;		
40 CALL LBL "RESET"		
41 M30		
42 ;		
43 * -	; Subprograms	
44 LBL "PLANE"		

45 TRANS DATUM AXIS X+0 Y+50 Z+0	; Shift the datum for tilted machining
46 PLANE SPATIAL SPA+2 SPB+0 SPC+0 TURN FMAX	; Tilt the working plane
47 LBL 0	
48 ;	
49 LBL "RESET"	
50 FUNCTION RESET TCPM	
51 M140 MB+50	
52 CALL PGM TNC:\nc_prog\SAFE.h	
53 TRANS DATUM RESET	
54 PLANE RESET TURN FMAX	
55 LBL 0	
56 END PGM 1438530 MM	

12

Compensations

12.1 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 215

Related topics

- Tool radius compensation
  Further information: "Tool radius compensation", Page 412
- Tool compensation with compensation tables
  Further information: "Tool compensation with compensation tables", Page 418

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: "Using TOOL CALL to call a tool", Page 205
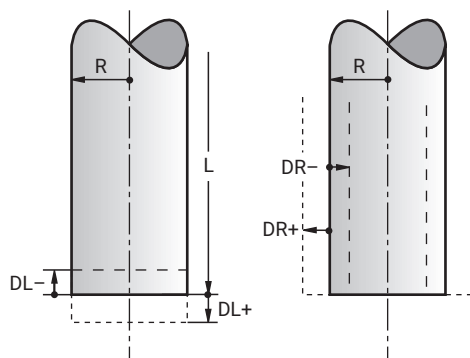
Delta values represent deviations from the length and radius of a tool.

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: "Using TOOL CALL to call a tool", Page 205

Further information: "Tool compensation with compensation tables", Page 418

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

Further information: "Tool data within variables", Page 410

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 = $\mathbf{L} + \mathbf{DL}_{TAB} + \mathbf{DL}_{Prog}$

L:	Tool length **L** from the tool table
DL $_{TAB}$:	Tool length delta value **DL** from the tool table
DL $_{Prog}$:	Tool length delta value **DL** from the tool call or the compensation table
	The most recently programmed value becomes active.
	Further information: "Using TOOL CALL to call a tool", Page 205
	Further information: "Tool compensation with compensation tables", Page 418

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:

- If tool radius compensation **RR** or **RL** is active
  Further information: "Tool radius compensation", Page 412
- 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 425

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

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

R:	Tool radius **R** from the tool table
	Further information: User's Manual for Setup and Program Run
DR $_{TAB}$:	Tool radius delta value **DR** from the tool table
	Further information: User's Manual for Setup and Program Run
DR $_{Prog}$:	Tool radius delta value **DR** from the tool call or the compensation table
	The most recently programmed value becomes active.
	Further information: "Using TOOL CALL to call a tool", Page 205
	Further information: "Tool compensation with compensation tables", Page 418

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 606

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 771

- 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 205

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

- The control takes up to six axes including the rotary axes into account in the tool compensation.

12.2 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 199

Requirements

■ Parameters have been defined in 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 = R + DR_{TAB} + DR_{Prog}

Further information: "Tool compensation for tool length and tool radius", Page 408

Paraxial traverses can be corrected 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
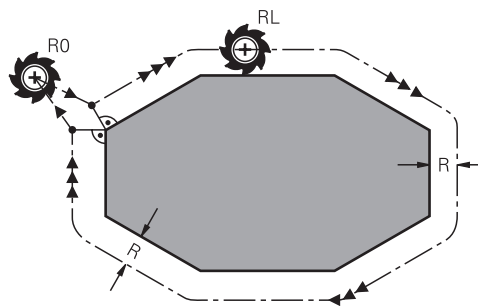
Radius-compensated traverse with path functions
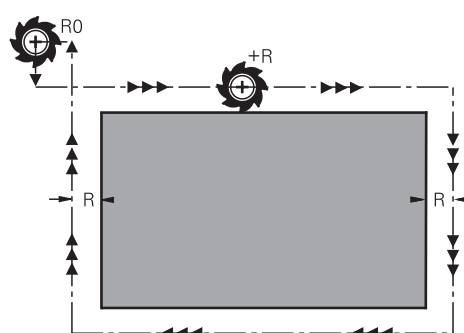
Radius-compensated traverse with paraxial movements

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.
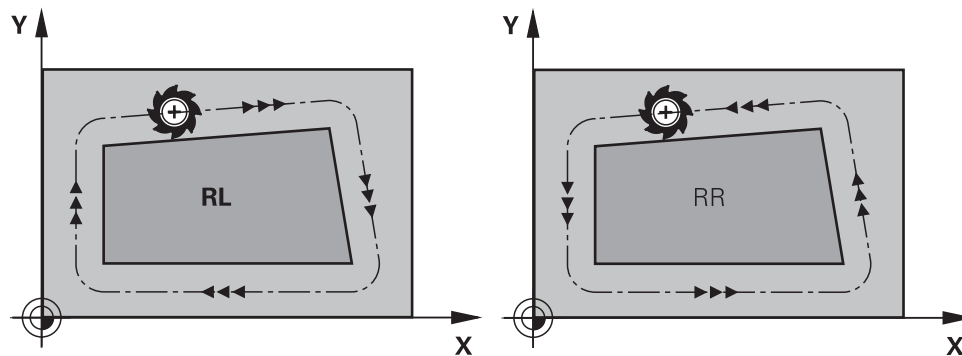
RL: The tool moves on the left of the contour

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.

> **i** 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
- The control takes up to six axes including the rotary axes into account in the tool compensation.
- 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 execute 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.

12.3 Tool radius compensation (TRC) with lathe tools (#50 / #4-03-1)

Application

The tip of a lathe tool has a certain radius **RS**. By default, programmed paths refer to the theoretical tool tip (i.e., the longest measured values ZL, XL and YL). When you machine tapers, chamfers and radii, the cutter radius **RS** causes deviations at the contour. The tool tip radius compensation prevents such deviations.

Related topics

- Tool data of turning tools
  Further information: User's Manual for Setup and Program Run
- Radius compensation with **RR** and **RL** in milling mode
  Further information: User's Manual for Setup and Program Run
- Presets on the tool
  Further information: "Presets on the tool", Page 199

Requirements

- Software option Turning (#50 / #4-03-1) or Adv. Spindle Interpol. (#96 / #7-04-1)
- Parameters have been defined for the tool type
  Further information: User's Manual for Setup and Program Run

Description of function

The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

In the turning cycles, the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.
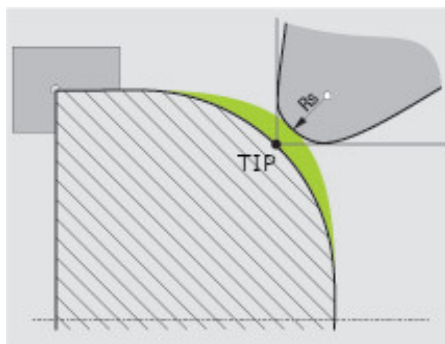
Offset between the tooth radius **RS** and the theoretical tool tip **TIP**

Theoretical tool tip TIP for tool radius compensation
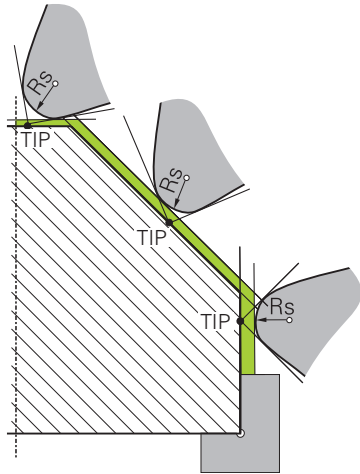
Inclined surface with theoretical tool tip **TIP** in the tool coordinate system **T-CS**

The theoretical tool tip is active in the tool coordinate system **T-CS**. The tool location point and the tool rotation point are at the tool tip.

Further information: "Tool coordinate system T-CS", Page 323

Further information: "Presets on the tool", Page 199
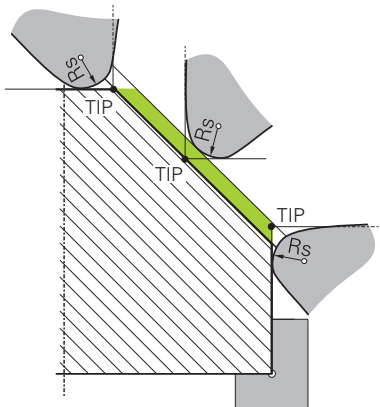
Inclined surface with theoretical tool tip **TIP** in the workpiece coordinate system **W-CS**

Only with the **FUNCTION TCPM** NC function with the **REFPNT TIP-CENTER** selection is the theoretical tool tip active in the workpiece coordinate system **W-CS**. The tool location point is at the tool tip. The tool rotation point is located at the tool center point.

Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

Further information: "Workpiece coordinate system W-CS", Page 316

Further information: "Presets on the tool", Page 199

Further information: "Simultaneous turning", Page 164

Notes
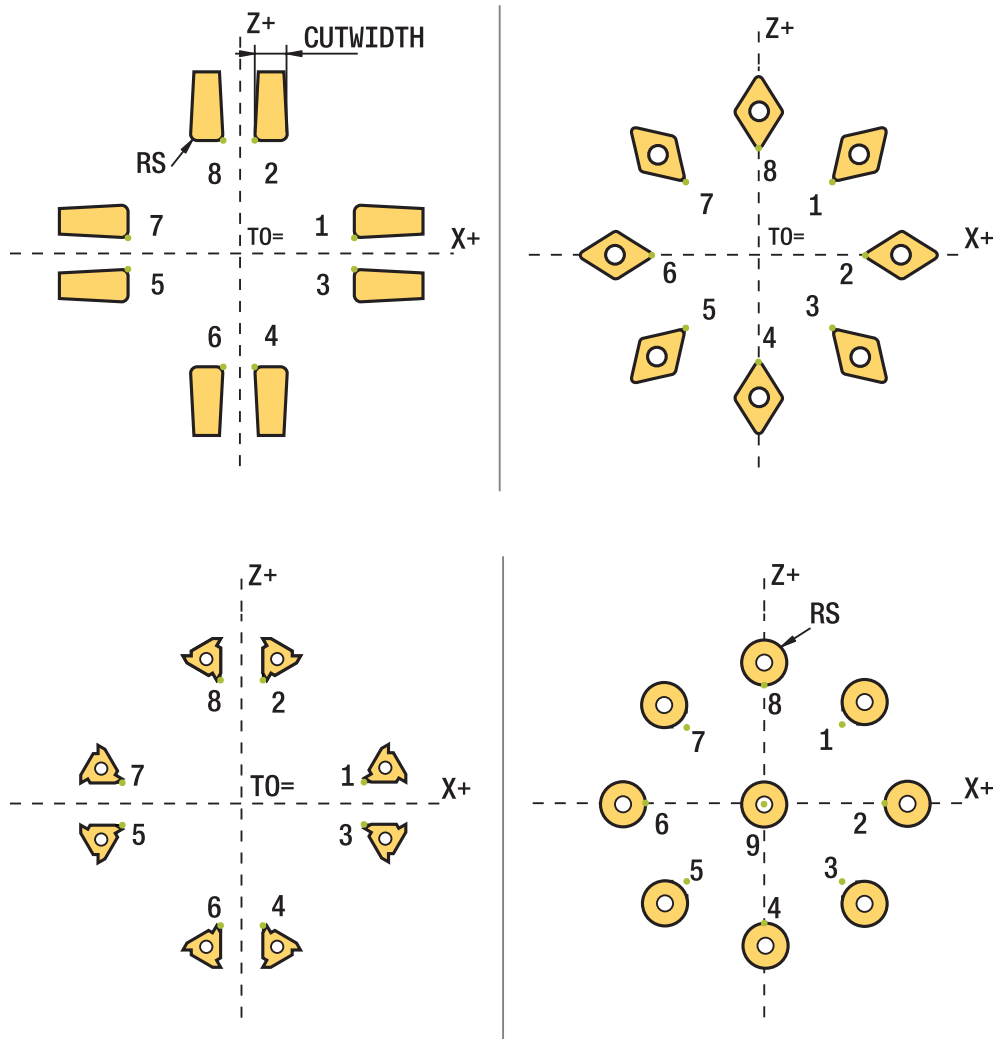
- The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.
- Tooth radius compensation is also possible during inclined machining.

  Active miscellaneous functions limit the possibilities here:
  - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
  - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tooth radius compensation with all positioning blocks (e.g., with **RL/RR**)
- The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress this warning with the machine parameter **suppressResMatlWar** (no. 201010).

12.4 Tool compensation with compensation tables

Application

With the compensation table, you can save compensations in the tool coordinate system (T-CS) or in the working plane coordinate system (WPL-CS). 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 310

Related topics

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 842

  Further information: "Compensation table *.wco", Page 844

- Editing compensation tables during program run

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

Description of function

In order to correct tool values by using the compensation tables, the steps below are needed:

- Creating a compensation table

  Further information: "The Create new table window", Page 811

- Activating the compensation table in the NC program

  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 420

- As an alternative, activating the compensation table manually for the program run

  Further information: "Activating the compensation tables manually", Page 420

- Activating a compensation value

  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 420

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

Further information: "Accessing table values ", Page 823

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 323

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**

  Further information: "Using TOOL CALL to call a tool", Page 205

- In the case of turning tools, as an alternative to **FUNCTION TURNDATA CORR-TCS** (#50 / #4-03-1)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)", Page 422

- In the case of grinding tools, as compensation for **LO** and **R-OVR** (#156 / #4-04-1)

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

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 318

The ***.wco** compensation tables are used mainly for turning (#50 / #4-03-1).

The compensations have the following effects:

- For turning operations, as an alternative to **FUNCTION TURNDATA CORR-WPL** (#50 / #4-03-1)
- An X shift affects the radius

The following options are available for a shift in the WPL-CS:

- **FUNCTION TURNDATA CORR-WPL**
- **FUNCTION CORRDATA WPL**
- Shifting with the turning-tool table
    - Optional **WPL-DX-DIAM** column
    - Optional **WPL-DZ** column

> **ⓘ** The shifts programmed with **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** are alternative programming options for the same shift.
>
> A shift in the working plane coordinate system (**WPL-CS**) defined by the turning-tool table is added to the **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** functions.

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**.

12.4.1   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 420

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 842

  Further information: "Compensation table *.wco", Page 844

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 **Parameter**	Path of table Text or string parameter Selection by means of a selection window

12.4.2   Activating a compensation value with FUNCTION CORRDATA

Application

The **FUNCTION CORRDATA** function allows activating a row of the compensation table for the active tool.

Related topics

- Selecting a compensation table

  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 420

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 842

  Further information: "Compensation table *.wco", Page 844

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 ▶ Special functions ▶ Functions ▶ Activating compensation values 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 Number, text, or variable 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

12.5 Compensating turning tools with FUNCTION TURNDATA CORR (#50 / #4-03-1)

Application

With **FUNCTION TURNDATA CORR** you can define additional compensation values for the active tool. In the **TURNDATA CORR FUNCTION** you can enter delta values for tool lengths in the X direction **DXL** and in the Z direction **DZL**. The compensation values have an additive effect on the compensation values from the turning tool table.

The compensation can be defined in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**.

Further information: "Reference systems", Page 310

Related topics

■ Delta values in the turning tool table
  Further information: User's Manual for Setup and Program Run
■ Tool compensation with compensation tables
  Further information: "Tool compensation with compensation tables", Page 418
■ Shaping with **FUNCTION SHAPING** (#96 / #7-04-1)
  Further information: "Workpiece shaping with FUNCTION SHAPING (#96 / #7-04-1)", Page 527

Requirements

■ Software option Turning (#50 / #4-03-1) or Adv. Spindle Interpol. (#96 / #7-04-1)
■ Parameters have been defined for the tool type
  Further information: User's Manual for Setup and Program Run

Description of function

The coordinate system in which the compensation is active can be defined:

■ **FUNCTION TURNDATA CORR-TCS**: Tool compensation is active in the tool coordinate system
■ **FUNCTION TURNDATA CORR-WPL**: Tool compensation is active in the workpiece coordinate system

With **FUNCTION TURNDATA CORR-TCS** you can define a cutter radius oversize **DRS**. This enables you to program an equidistant contour oversize. **DCW** allows you to correct the recessing width of a recessing tool.

Tool compensation **FUNCTION TURNDATA CORR-TCS** is always active in the tool coordinate system, even during inclined machining.

FUNCTION TURNDATA CORR is always effective for the active tool. A renewed **TOOL CALL** deactivates compensation again. When you exit the NC program, the control automatically resets the compensation values.

Input

11 FUNCTION TURNDATA CORR-TCS:Z/X DZL:+0.1 DXL:+0.05 DCW:+0.1	; Tool compensation in Z direction, X direction and for the width of the recessing tool

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Turning tool compensation TURNDATA

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA CORR	Syntax initiator for tool compensation of a turning tool
CORR-TCS:Z/X or **CORR-WPL:Z/X**	Tool compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**
DZL:	Delta value for the tool length in Z direction Optional syntax element
DXL: or **DXL-DIAM:**	Delta value for the tool length in X direction, given as a radius or diameter value **DXL-DIAM:** only if **CORR-WPL:Z/X** has been selected Optional syntax element
DCW:	Delta value for the recessing tool width Only if **CORR-TCS:Z/X** was selected Optional syntax element
DRS:	Delta value for the cutter radius Only if **CORR-TCS:Z/X** was selected Optional syntax element

Note

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.

The values of the function **FUNCTION TURNDATA CORR** take the effect of delta values from the NC program.

Note in connection with the interpolation turning (#96 / #7-04-1)

During interpolation turning, the functions **FUNCTION TURNDATA CORR** and **FUNCTION TURNDATA CORR-TCS** are not active.

If you want to compensate for a turning tool in Cycle **292 CONTOUR.TURNG.INTRP.**, compensation needs to be performed in the cycle or in the tool table.

Further information: User's Manual for Machining Cycles

12.6   3D tool compensation (#9 / #4-01-1)

12.6.1   Fundamentals

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

Further information: "Straight line LN", Page 425

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 427

Here are some examples of where 3D tool compensation can be used:

- 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:
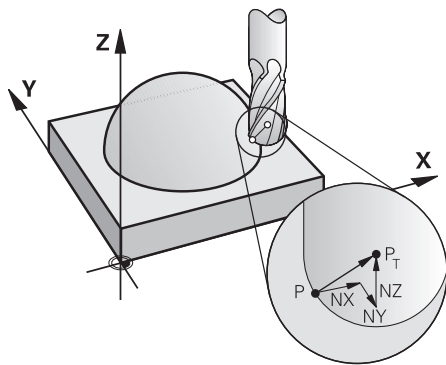
> ℹ️ For an optional tool angle of inclination, the NC blocks must include an additional tool vector with the components TX, TY and TZ.
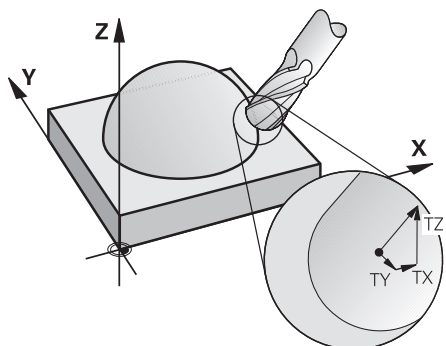
> ℹ️ Note the differences between face milling and peripheral milling.
>
> **Further information:** "3D tool compensation during face milling (#9 / #4-01-1)", Page 428
>
> **Further information:** "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 435

12.6.2 Straight line LN

Application

Straight lines **LN** are a prerequisite for 3D compensation. Within straight lines **LN**, a surface normal vector defines the direction of the 3D tool compensation. An optional tool vector defines the tool angle of inclination.

Related topics

- Fundamentals of 3D compensation

  Further information: "Fundamentals", Page 424

Requirements

- Software option Adv. Function Set 2 (#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 538

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 224

In addition, the straight lines **LN** contain a surface normal vector as well as an optional tool vector.

Input

> **LN X+31.737 Y+21.954 Z+33.165 NX+0.2637581 NY+0.0078922 NZ-0.8764339 TX +0.0078922 TY-0.8764339 TZ+0.2590319 F1000 M128**

The NC function includes the following syntax elements:

Syntax element	Meaning
LN	Syntax initiator for straight line with vectors
X, **Y**, **Z**	Coordinates of the straight-line end point
NX, **NY**, **NZ**	Components of the surface normal vector Optional syntax element
TX, **TY**, **TZ**	Components of the tool vector Optional syntax element, only effective in combination with **FUNCTION TCPM** or **M128**
R0, **RL** or **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 412 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.
- Make sure to always program all three vector components, as the control will not take over any values 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.

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

12.6.3 Tools for 3D tool compensation

Application

3D tool compensation can be used with the following tool shapes: end mill, toroid cutter and ball-nose cutter.

Related topics

- Compensation in tool management

  Further information: "Tool compensation for tool length and tool radius", Page 408

- Compensation in tool call

  Further information: "Using TOOL CALL to call a tool", Page 205

- Compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 418

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 for the tool. The surface normal vector always points to the tool radius 2 center CR2.
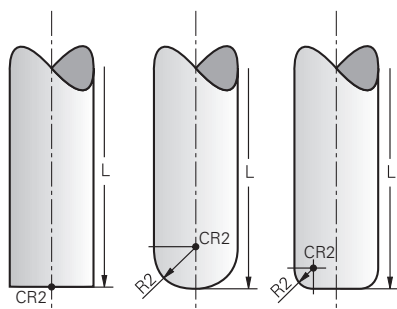
Position of CR2 with the individual tool shapes

Further information: "Presets on the tool", Page 199

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 using the overall length and outputting an NC program referring 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 593

  Use the simulation to ensure that no contours are damaged by the tool oversize.

12.6.4 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.
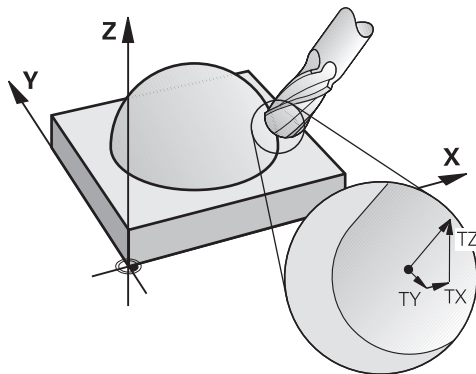
Requirements

- Adv. Function Set 2 (#9 / #4-01-1) software option
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 425

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

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 576

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

Description of function

The variants below are possible with face milling:

- **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
- **LN** block without tool orientation **T**, but with a surface-normal vector **N**, with **M128**, or **FUNCTION TCPM** active: The control interprets the surface-normal vector **N** as the tool vector **T**, too, and approaches the tool perpendicularly to the workpiece contour. For safety reasons, HEIDENHAIN does not recommend this kind of programming.

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 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

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

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.
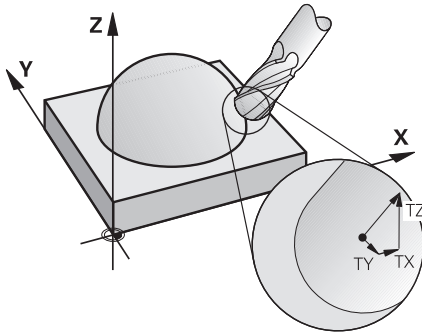
- If a tool orientation **T** has been defined in the **LN** block and **M128** (or **FUNCTION TCPM**) is active at the same time, then the control will position the rotary axes automatically in such a way that the tool can reach the specified tool orientation. If you have not activated **M128** (or **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 438

Examples

**Compensate re-worked ball-nose cutter
CAM output at tool tip**
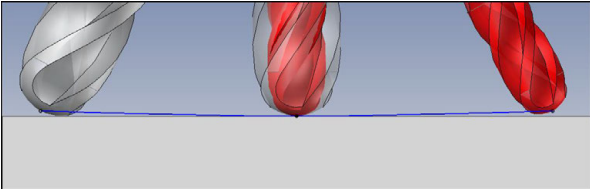
Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

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

Proposed solution:

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

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

Compensate re-worked ball-nose cutter CAM output at the center of the ball
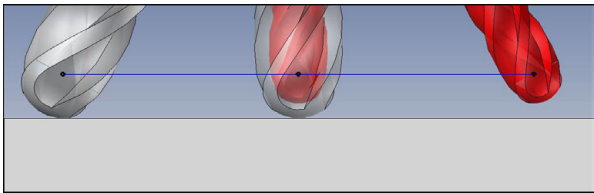
Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

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

Suggested solution:

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

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

> **i** With TCPM **REFPNT CNT-CNT** the tool compensation values are identical for the outputs on the tool tip or center of the ball.

Create workpiece oversize
CAM output at tool tip

Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

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

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
    - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

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

Create workpiece oversize
CAM output at the center of the ball
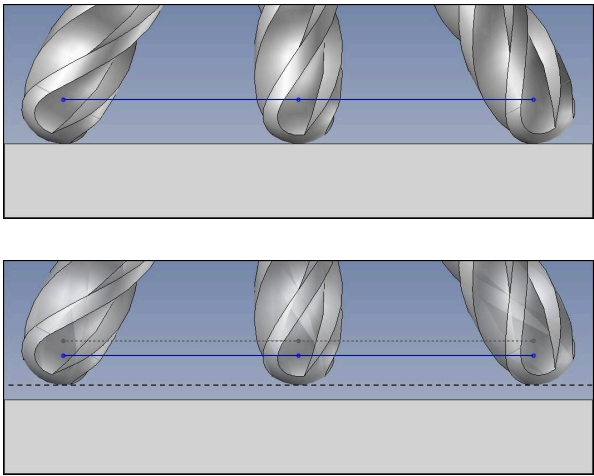
Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the ball
- TCPM function **REFPNT CNT-CNT**
- Vector program with surface normal vectors and tool vectors

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
    - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

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

12.6.5 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.
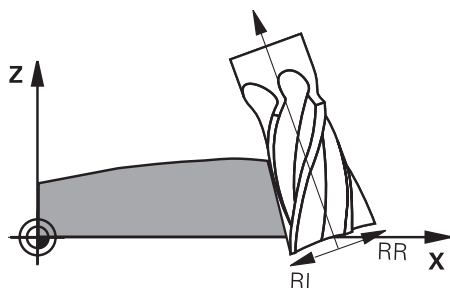
Requirements

- Adv. Function Set 2 (#9 / #4-01-1) software option
- Machine with automatically positionable rotary axes
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)", Page 576

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

- NC program with tool radius compensation **RL** or **RR**

  Further information: "Tool radius compensation", Page 412

Description of function

The variants below are possible with peripheral milling:

- **L** block with or without programmed rotary axes, **M128** or **FUNCTION TCPM** is active, define compensation direction with radius compensation **RL** or **RR**
- **LN** block with tool orientation **T** without N vector, **M128**, or **FUNCTION TCPM** is active

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 TX-0.0807 TY-0.3409 TZ0.9366 RR M128	; Compensation is possible, compensation direction RR

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

- 3D tool compensation can be used in NC programs for peripheral milling with spatial or axis angles. It is also possible to use vector programs with tool vectors or NC programs without tool inclination.

- If you combine vector programs with surface-normal vectors and tool vectors with **RL** or **RR**, the control will ignore the surface-normal vectors.

- 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 438

Example

Compensate re-worked end mill
CAM output at tool center
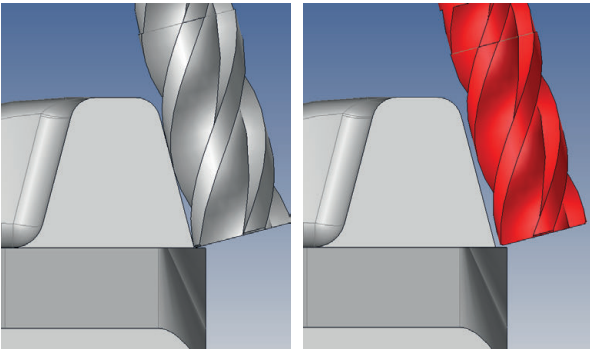
You use a re-worked Ø 11.8 mm end mill instead of Ø 12 mm.

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 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

12.6.6 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 424

■ Tools for 3D compensation

   Further information: "Tools for 3D tool compensation", Page 427

Requirements

■ Software option Adv. Function Set 2 (#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 538

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

12.7 3D radius compensation depending on the tool contact angle (#92 / #2-02-1)

Application

Due to the production process, the effective spherical radius of a ball cutter deviates from the ideal form. The maximum form inaccuracy is defined by the tool manufacturer. Common deviations lie between 0.005 mm and 0.01 mm.

The form inaccuracy can be saved in the form of a compensation-value table. This table contains angle values and the deviation from the nominal radius **R2** measured on the respective angle value.

The 3D-ToolComp (#92 / #2-02-1) software option enables the control to correct the value defined in the compensation value table depending on the actual contact point of the tool.

3D calibration of the touch probe can also be carried out with the 3D-ToolComp software option. During this process the deviations determined during touch probe calibration are saved to the compensation value table.
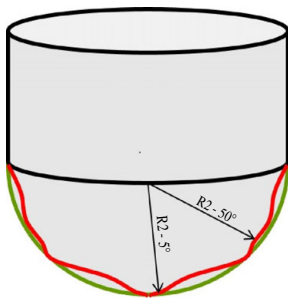
Related topics

- Compensation value table *.3DTC

  Further information: "*.3DTC compensation table ", Page 845
- Touch probe 3D calibration

  Further information: User's Manual for Setup and Program Run
- 3D probing with a touch probe

  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- 3D compensation with CAM-generated NC programs with surface-normal vectors

  Further information: "3D tool compensation (#9 / #4-01-1)", Page 424

Requirements

- Adv. Function Set 2 (#9 / #4-01-1) software option
- 3D-ToolComp (#92 / #2-02-1) software option
- Output of surface normal vectors from the CAM system
- The tool has been defined appropriately in the tool management:
    - Value of 0 in the column **DR2**
    - Name of the matching compensation table in the column **DR2TABLE**

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

Description of function
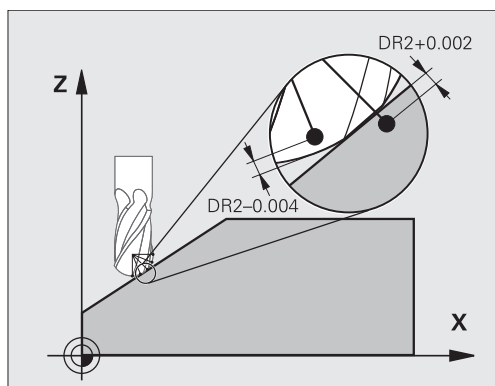
If you are executing an NC program with surface-normal vectors and have assigned a compensation value table (DR2TABLE column) to the active tool in the tool table (TOOL.T), the control uses the values from the compensation value table instead of the compensation value DR2 from TOOL.T.

In doing so, the control takes the compensation value from the compensation value table defined for the current contact point of the tool with workpiece into account. If the contact point is between two compensation points, the control interpolates the compensation value linearly between the two closest angles.

Angle value	Compensation value
40°	0.03 mm (measured)
50°	-0.02 mm (measured)
45° (contact point)	+0.005 mm (interpolated)
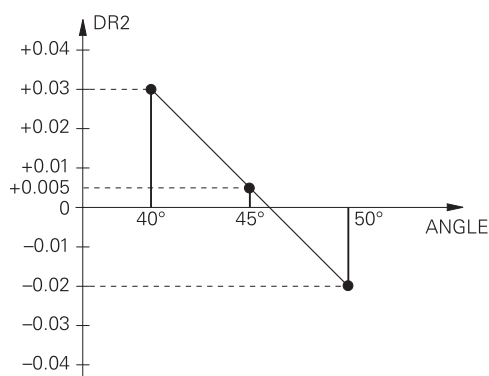
Notes

- If the control cannot interpolate a compensation value, it displays an error message.
- **M107** (suppress error message for positive compensation values) is not required, even if positive compensation values are determined.
- The control uses either DR2 from TOOL.T or a compensation value from the compensation value table. Additional offsets, such as a surface oversize, can be defined via DR2 in the NC program (compensation table **.tco** or **TOOL CALL** block).

13

Files

13.1 File management

13.1.1 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, buttons and shortcuts

The file management provides the following icons, buttons and shortcuts:

Icon, button or shortcut	Meaning
▽	**Activate custom filter** **Further information:** "Creating or changing a user-defined filter", Page 450 Only in the **Files** operating mode
←	**Back**
∨	Open or close **History** The control opens a selection menu with up to 20 previous paths since switch-on.
✎ ALT + O	**Edit** The control displays a navigation path as editable text.
→ ENT	**Finish editing** The control saves the changes to the path and calls the new path.
↻	**Refresh**
★	Favorite If you add a favorite, then the control displays this icon next to the file or the folder.
⏏	**Eject** Eject USB device
<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
	ℹ You create a new table in the **Tables** operating mode. **Further information:** "The Tables operating mode", Page 808
File functions	The control opens the context menu. **Further information:** "Context menu", Page 747 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 454 Only in the **Files** operating mode
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 and selects the first NC block. Only in the **Files** operating mode
Update TAB / PGM	Converting the format and content of files from the iTNC 530 Modify faulty files **Further information:** "Adapting files", Page 456 Only in the **Files** operating mode
Mount network share	**Further information:** User's Manual for Setup and Program Run Only in the **Files** operating mode

If you swipe a file or folder to the right in the file management, the control displays the following file functions:

Icon	Meaning
⊏⨉⊐	**Rename**
⧉	**Copy**
✂	**Cut** If you cut a file or a folder, then the control dims the icon of the file or the folder.
🗑	**Delete**

Icon	Meaning
🔒	**Activate write-protection** If write-protection is active, then the control displays this icon next to the file or the folder.
🔓	**Deactivate write-protection**
☆⁺	**Add favorite**
☆⁻	**Remove favorite**

You can also select some of these file functions from the context menu.

Further information: "Context menu", Page 747

Areas of file management
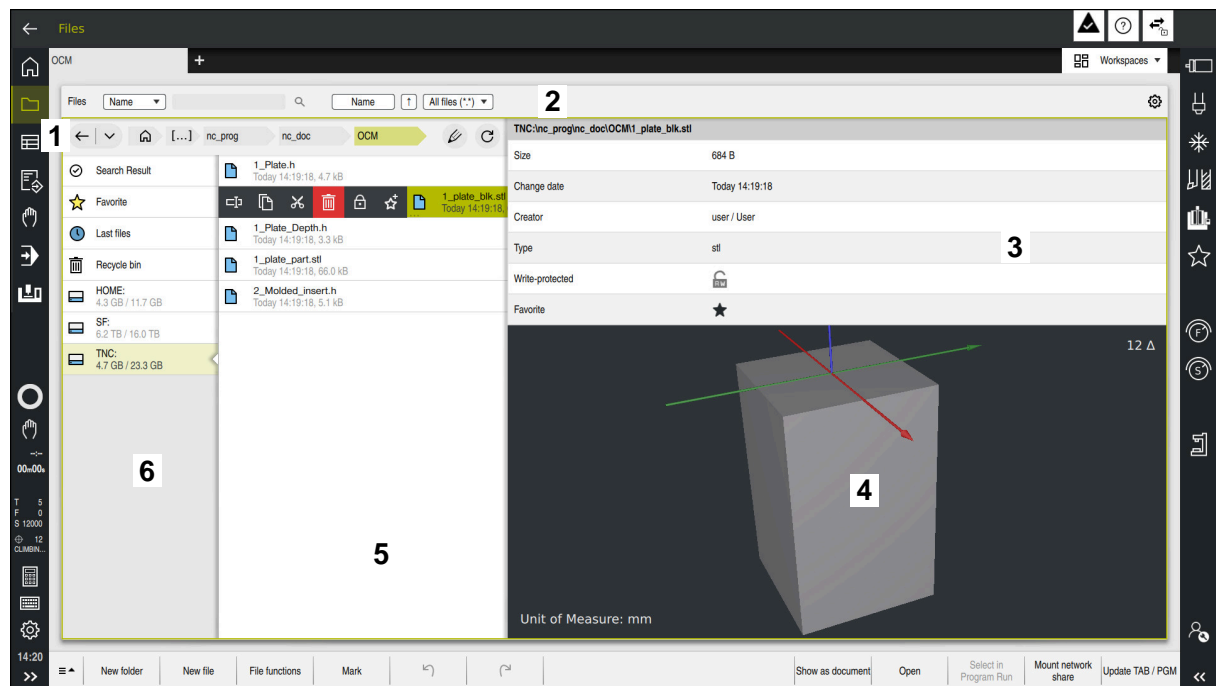
The **Files** operating mode

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. You can edit the path or open a previous path from the History.

2   Title bar

■   Full-text search

Further information: "Full-text search in the title bar", Page 446

■   Sorting

Further information: "Sorting in the title bar", Page 446

■   Filter

Further information: "Filters in the title bar", Page 446

■   Settings

Further information: "Settings in the title bar", Page 446

3   Information area

Further information: "Information area", Page 446

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

6   Navigation column

Further information: "Navigation column", Page 447

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.

Filters in the title bar

The control provides standard filters and a user-defined filter that can be used to filter by the desired file types.

Further information: "Creating or changing a user-defined filter", Page 450

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 447

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 747

- 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 132
I	NC program with ISO commands
HC	Contour definition in the smarT.NC format of the iTNC 530
HU	Main program in the smarT.NC format of the iTNC 530
3DTC	Table with 3D tool compensations that are independent of the tool angle (#92 / #2-02-1) **Further information:** "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439
D	Table with workpiece datums **Further information:** "Datum table *.d", Page 832
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 789
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) **Further information:** "Point table *.pnt", Page 830
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 827 **Further information:** "Cutting data calculator", Page 756
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 **Further information:** User's Manual for Setup and Program Run
TRN	Table with turning tools (#50 / #4-03-1) **Further information:** User's Manual for Setup and Program Run

File type	Application
GRD	Table with grinding tools (#156 / #4-04-1)
	Further information: User's Manual for Setup and Program Run
DRS	Table with dressing tools (#156 / #4-04-1)
	Further information: User's Manual for Setup and Program Run
TNCDRW	Contour description as a 2D drawing
	Further information: "The Contour graphics workspace ", Page 675
M3D	Format for tool carriers or collision objects (#40 / #5-03-1), for example
	Further information: "Options for fixture files", Page 476
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 774
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 476
	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

File type	Application
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 51
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
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

Creating or changing a user-defined filter

To create or edit a user-defined filter in file management:

▶ Select the **Files** operating mode, if required
▶ Open the standard filters selection menu
▶ Select **User-defined**
> The control displays an input field next to the selection menu.
▶ Enter the desired file types in the input field (e.g., ***.h, *.txt**)

▽
▶ Confirm your input and activate the filter
> The control displays all files of the specified file types and takes over the text to the selection menu.

ⓘ The user-defined filter will remain active only until you shut down the control.

Further information: "Filters in the title bar", Page 446

Notes

- The control has 189 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 808
- 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 655
- 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 provides 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.

13.1.2 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	Meaning
✚	**Add** in the **Tables** and **Editor** operating modes
📄	**Open File** in the **Program Run** operating mode

Icons and buttons

The **Open File** workspace provides the following icons and buttons:

Icon or button	Meaning
▢	Show or hide the **File preview**
New folder	Create new folder Only in the **Tables** and **Editor** operating modes
New file	Create new file Only in the **Tables** and **Editor** operating modes
Use in simulation	Use the table for simulation Only in the **Editor** operating mode if tool tables, preset tables, or datum tables have been selected
Open	Open the selected file

13.1.3 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**

■ **Editor**

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 811
- **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**

In both areas, the control displays the **Tool management**, **Datums**, and **Presets** buttons. They will open the table that is active in program run or in the simulation. The control displays the **M** or **S** status on the application tab.

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 132
New graphical programming	**Contour** **Further information:** "The Contour graphics workspace ", Page 675
New text file	- **Text file** with the ***.txt** extension - **Format file** with the ***.a** extension **Further information:** "The Text editor workspace", Page 459
New job	**Job list** **Further information:** "The Job list workspace", Page 789

13.1.4 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 448
- **Show as document** button in the **Files** operating mode
  Further information: "Icons, buttons and shortcuts", Page 442

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 78

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 (e.g., *webm)

Further information: "File types", Page 448

You can, for example, transfer dimensions from a technical drawing using the clipboard in the NC program.

In the **Document** workspace, the control provides a context menu with additional functions for every file type (e.g., navigating back within the opened files).
Further information: "Context menu in the Document workspace", Page 752

Icons in the Document workspace

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

Icon	Meaning
	Open File
	Further information: "Open file", Page 455
	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
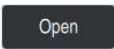
Open

▶ Select **Open**

> The control displays the file in the **Document** workspace.

13.1.5 Adapting files

Application

There are certain files that the TNC7 cannot open or display correctly (e.g., files from the iTNC 530 or files edited in a text editor). These files can be adapted (i.e., converted with the **Update TAB / PGM** function to suit the control's requirements).

Description of function

NC programs

The control uses the **Update TAB / PGM** function to remove umlauts and checks whether the NC block **END PGM** exists. Without **END PGM**, the NC program is not complete.

Tables

If you convert tables from an earlier control using the **Update TAB / PGM** function, then the control makes the following changes as needed:

- Changing the decimal separator from a comma to a point
- Taking over all supported tool types and defining all unknown tool types as **Undefined**

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 required, you can use the **Update TAB / PGM** function to adapt tables from the TNC7 or tables from previous software versions.

The changes that will be made include the following:

- Missing table columns are added (e.g., in tables from previous software versions)
- The character encoding of text columns is changed to UTF-8 (for tables with German umlauts or special characters)

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

Adapting a file

Prepare a backup of the original file before adapting

To adapt (convert) a file (format and contents):

📁
- ▶ Select the **Files** operating mode

- ▶ Select the desired file

Update TAB / PGM
- ▶ Select **Update TAB / PGM**
- ▷ The control adapts the file format and content.

> ℹ️ 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

- Using the **Update TAB / PGM** function, you can adapt all files in an entire folder at the same time.
- 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.

13.1.6 USB devices

Application

A USB device allows transmitting data and saving data externally.

Requirements

- 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

  > ℹ 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
- USB devices are permitted in **SELinux**
  Further information: User's Manual for Setup and Program Run
- If user administration is active, the HEROS.NormalUser role
  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.
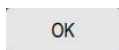
OK | ▶ Press **OK**
   | > The control shows the message **The USB device can be removed now**.

Notes

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

▶ Copy the NC program and all called files to the **TNC:** drive

---

NOTICE

Caution: Data may be lost!

Always remove a connected USB device properly, otherwise data may be damaged or deleted!

▶ Use the USB port for transfer and backup only; do not use it for editing and executing NC programs
▶ Use the icon to remove USB devices when data transfer is complete

---

- If an error message is displayed when connecting a USB device, check the setting in the **SELinux** security software.
  Further information: 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

13.2 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 448
■ Displaying text files in the **Document** workspace
Further information: "The Document workspace", Page 454

Description of function

The **Text editor** workspace is available in the **Editor** operating mode.

By default, the control opens the following file types in the **Text editor** workspace:

■ Text files, such as ***.txt**
Example: measuring logs output with **FN 16**
■ Format files, such as ***.a**
Example: format file for **FN 16**

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 622

> **i** Refer to your machine manual.
> In the machine parameter **standardEditor** (no. 102902), the machine manufacturer defines the default editor to be used for the file types (e.g., the **Text editor**).

Select the **Open in text editor** function in the context menu of the **Files** operating mode to open any file type in the **Text editor** workspace.

Further information: "Context menu in the Files operating mode", Page 748

Further information: "File types", Page 448

Icons in the Text editor workspace

The following icons are shown in the **Text editor** workspace:

Icon or shortcut	Meaning
$\mathbf{Q}$ **CTRL + F**	Open and close the **Search** column **Further information:** "The Search column in the Program and Text editor workspaces", Page 741
☐	Show or hide the **Line number**
⇥	Activate or deactivate the **Line break** When activating the **Line break**, the control will automatically add line breaks in the text. The control adapts the text length to the workspace size. The text is not truncated, but wraps to the next line, as indicated by the missing line number in the first line.
¶	Show or hide the **Formatting characters** If you show formatting characters, the control will replace blanks with symbols, such as dots, for the space characters.

If you show formatting characters, the control will display the following icons:

Icon	Meaning
. . .	**Blank space**
⟶	**Tabulator** When you press the **TAB** key during text entry, the control will insert a blank up to the next tab stop (the maximum is a whole tab width). The tab width can be defined in the editor settings.
LF	**End of line** The control indicates the line end by an **LF** symbol. If **Line break** is active, the text contents of a text line may be distributed to several lines, depending on the workspace width. This symbol indicates the actual end of the text line.

> ⓘ The control also supports the **CR LF** or **CR** line ends, which might be included in transmitted files.
> The line break behavior is always identical, independent of which symbol is displayed.

NC editor settings

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

The General information area

The **General information** area contains the following settings:

Setting	Meaning
Automatic saving	Save changes made to the file automatically or manually
	If this toggle switch is active, the control will save the file automatically when you do the following:
	■ Close the file
	■ Navigate to another tab
	■ Activate another 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.
Replace tabs with blanks	If this toggle switch is active, the control will insert a predefined number of blanks instead of a tab character.
Tab width	Number of characters or number of blanks for a tab until the tab stop

i If you change the width, the control will also change the width of the existing tabs.

If you inserted blanks instead of a tab character, the control will not update their number after a change of the width.

Notes

■ If you mark characters through touch operation, the control displays two marker symbols below the cursor.

■ In order to avoid undesired edits to a file, open it in the **Document** workspace.
Further information: "The Document workspace", Page 454

13.3   Programmable file functions

Application

Programmable file functions enable management of files from within the NC program. Files can be opened, copied, relocated, and deleted. This allows, for example, opening a component drawing 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 448

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

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

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

> **i**   In order to avoid program run interruptions or having to select an alternative HEROS tool, open a file of the corresponding file type once in the file manager. If the files of a certain file type can be opened in multiple HEROS tools, you can use the file manager to select the HEROS tool to be used for opening files of this file type.
>
> **Further information:** "File management", Page 442

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 ▶ File functions 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 ▶ File functions 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 ▶ File functions 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 selection 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.

14

Collision monitoring

14.1 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.
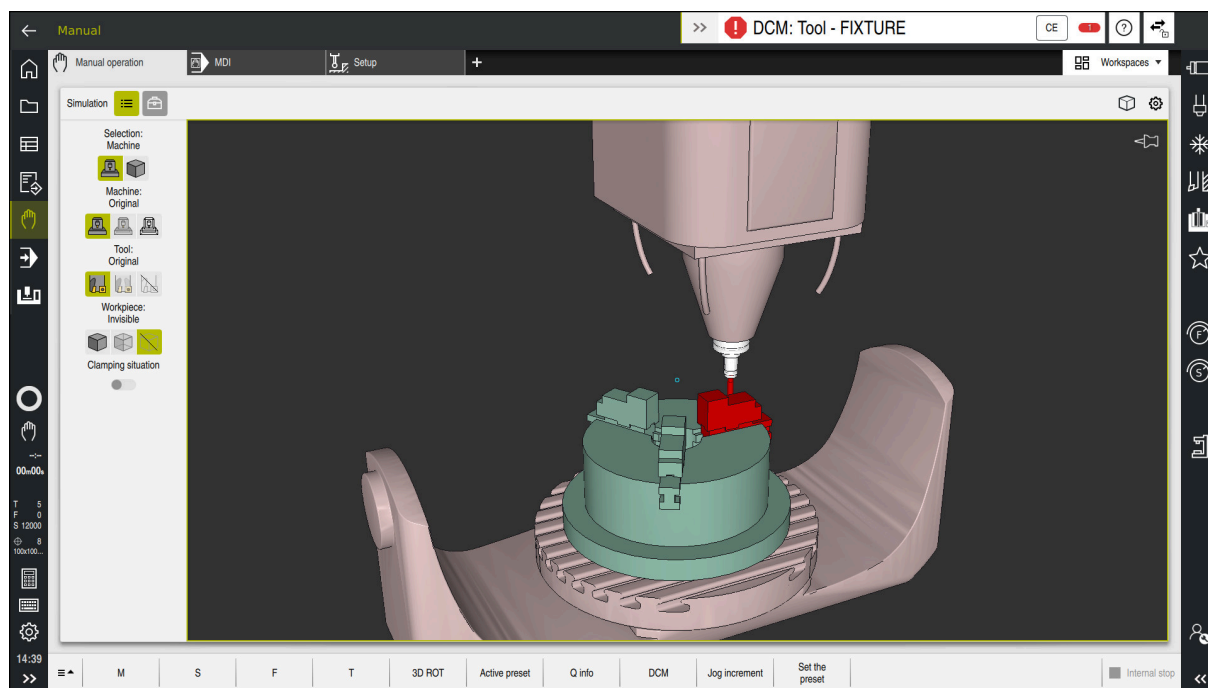
Dynamic Collision Monitoring (DCM) including collision warning

Related topics

■ Fundamentals of fixture management

   Further information: "Fixture management", Page 475

■ Extended tests in the simulation

   Further information: "Advanced checks in the simulation", Page 481

■ 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 479

Requirements

- Software option Collision Monitoring (#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 475
- 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.

>> 🛑 DCM: Tool - FIXTURE   CE
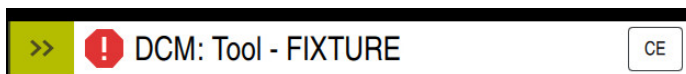
Error message for Dynamic Collision Monitoring (DCM)

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk of collision during all movements!

- ▶ Make sure to activate DCM whenever possible
- ▶ Make sure to always re-activate DCM immediately after a temporary deactivation
- ▶ Carefully test your NC program or program section in **Single Block** mode while DCM is deactivated

---

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

> ### *NOTICE*
>
> **Danger of collision!**
>
> Even if Dynamic Collision Monitoring (DCM) is active, the control will not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!
>
> ▶ Activate the **Advanced checks** toggle switch for the simulation
> ▶ Check the machining sequence using a simulation
> ▶ Carefully test your NC program or program section in the **Single Block** mode

Further information: "Advanced checks in the simulation", Page 481

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.

> ℹ 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 472

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

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

---

i If the **Advanced checks** function is activated, the control will display collisions during simulation (e.g., between workpiece and tool).

Further information: "Advanced checks in the simulation", Page 481

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 764

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the tool change position in the machine.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Global program settings (GPS) (#44 / #1-06-1) are not available
- Handwheel override 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.

---

i The control displays the status of Dynamic Collision Monitoring (DCM) in the **Simulation** workspace.

Further information: "Icons in the Simulation workspace", Page 763

Activating the graphic display of the collision objects
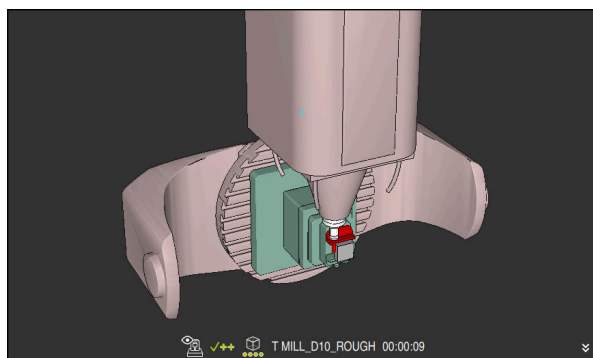
Simulation in the **Machine** mode

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

NOTICE
Danger of collision!
Dynamic Collision Monitoring DCM does not detect collisions caused by the reciprocating stroke. Risk of collision!
▶ Carefully prove-out the NC program

- 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.

14.1.1 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 479

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 possible, for example, 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:

<div style="display:inline-block; border:1px solid #000; padding:4px; font-size:small;">Insert
NC function</div>

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **FUNCTION DCM**
- ▶ Select the **OFF** or **ON** syntax element

14.2 Fixture management

14.2.1 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 468
- Integrating an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 193

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 476
- 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 478
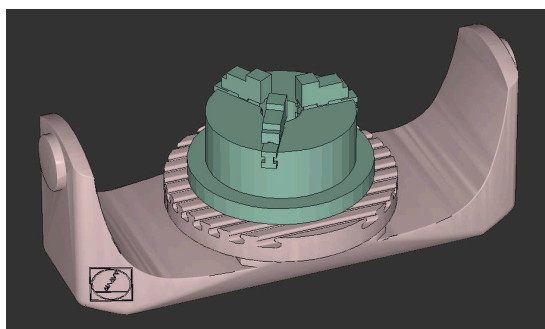
Three-jaw chuck loaded as fixture

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 **3D mesh** function (#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 CAD Model Optimizer software option (#152 / #1-04-1) you can adapt STL files that do not meet the requirements and then use them as 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.
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

  Further information: "Advanced checks in the simulation", Page 481

14.2.2 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 475

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 ▶ Fixtures (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.

14.2.3 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 468
- Loading and removing the fixture
  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 478

Requirements

- Software option Collision Monitoring v2 (#140 / #5-03-2)
- Dynamic Collision Monitoring (DCM) is active
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 468
- Fixture is integrated in the NC program
  Further information: "Load and remove fixtures with the FIXTURE NC function", Page 478

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 initiator 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 Number or numerical parameter 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, prove-out the affected NC program sections in **Single Block** mode

The control cannot approach the reduced minimum clearance with the **RESTORE POSITION** function. If the approach position falls short of 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

14.3 Advanced checks in the simulation

Application

The **Advanced checks** function allows you to check in the **Simulation** workspace whether collisions will occur (e.g., between the workpiece and the tool). This avoids unplanned downtimes due to a collision.

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 468

Description of function

The **Advanced checks** function can be used only in the **Editor** operating mode.

If you activate the **Advanced checks** toggle switch in the **Visualization options** column, the control opens the **Advanced checks** window.

Further information: "The Visualization options column", Page 764

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.

*   **Collision between workpiece and tool**

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

*   **Collision between workpiece and machine**

    The control displays a warning in case of collisions between the workpiece and the machine (e.g., spindle).

    The control does not consider the tool and the workpiece fixture.

*   **Fixture collision**

    The control displays a warning in case of collisions between the workpiece fixture and the tool including the tool carrier.

The control displays the material removal at rapid traverse and collision objects in red.

The control also considers inactive steps of a stepped tool.

You can activate several tests at the same time.

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 186

14.4 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 588

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

  Further information: "Retracting in the tool axis with M140", Page 584

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

  This makes sense for turning operations (#50 / #4-03-1)

- **FUNCTION LIFTOFF RESET**: NC function reset

Further information: "Tool coordinate system T-CS", Page 323

The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

FUNCTION LIFTOFF in turning mode (#50 / #4-03-1)

> *NOTICE*
>
> **Caution: Danger to the tool and workpiece!**
>
> Undesired movements of the axes can occur if you use the **FUNCTION LIFTOFF ANGLE TCS** function in turning mode. The behavior of the control depends on the kinematics description and Cycle **800** (**Q498 = 1**).
>
> ▶ Carefully test the NC program or program section in **Single Block** mode
> ▶ If necessary, change the algebraic sign of the defined angle

If parameter **Q498** has been set to 1, the control will reverse the tool for machining.

In conjunction with the **LIFTOFF** function, the control behaves as follows:

- If the tool spindle has been defined as an axis, the **LIFTOFF** direction will be reversed.
- If the tool spindle has been defined as a kinematic transformation, the **LIFTOFF** direction will not be reversed.

Further information: User's Manual for Machining Cycles

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** ▶ **Tool retraction 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 468

- 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.

15

Control Functions

15.1 Adaptive Feed Control (AFC) (#45 / #2-31-1)

15.1.1 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 Contr. (#45 / #2-31-1) software option
■ 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 489

■ 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 490

■ 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.
- During turning (#50 / #4-03-1), the control can monitor only tool wear and tool load, but cannot influence the feed rate.

  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 445

15.1.2 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 Contr. (#45 / #2-31-1) software option
- 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 490

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

> ### *NOTICE*
>
> **Caution: Danger to the tool and workpiece!**
>
> If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!
>
> ▶ Program the **FUNCTION MODE TURN** machining mode before the tool call

- The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.
- Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.
- You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

  If you program both values, the control will use the value programmed in the NC program!

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 489

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- 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 regulates the feed rate override function.
  - Using the potentiometer to increase the feed-rate override has no influence on the controlling process.
  - 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 the controlling process 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.

15.2 Functions for controlling program run

15.2.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION S-PULSE	Program pulsing spindle speed	Page 492
FUNCTION DWELL	Program singular dwell time	Page 493
FUNCTION FEED DWELL	Program cyclic dwell time	Page 494

15.2.2 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 when turning at a constant speed (#50 / #4-03-1), 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	Syntax initiator 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.

15.2.3 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 494

Description of function

The defined dwell time from **FUNCTION DWELL** is effective in both milling and turning mode.

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

15.2.4   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 in a turning cycle.

Related topics

- Program a one-time dwell time

  Further information: "Programmed dwell time with FUNCTION DWELL", Page 493

Description of function

The defined dwell time from **FUNCTION FEED DWELL** is effective in both milling and turning mode.

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 ▶ Dwell time FEED/DWELL ▶ 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**.

16

Monitoring

16.1 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.

> **i** If process monitoring (#168 / #5-01-1) displays a process heat map in the simulation, the control does not display a component heat map.
> **Further information:** "Process monitoring (#168 / #5-01-1)", Page 501

Related topics

- The **COMPMON** tab in 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 766
- **Process Monitoring** (#168 / #5-01-1) with **SECTION MONITORING**
  Further information: "Process monitoring (#168 / #5-01-1)", Page 501

Requirements

- Software option Component Monitoring (#155 / #5-02-1)
- Components to be monitored are defined
  In the optional machine parameter **CfgMonComponent** (no. 130900), the machine manufacturer defines the machine components to be monitored as well as the warning and error thresholds.

Description of function

A component heatmap is similar to the image from an infrared camera.

The heatmap displays a color image consisting of the following basic colors:

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

In addition, the control displays the following colors:

- Light gray: no component was configured
- Dark gray: component cannot be monitored (e.g., due to incorrect or missing details within the configuration)

> **i** Refer to your machine manual.
> The machine manufacturer configures the components.

The control shows these statuses on the workpiece in the simulation and can overwrite the statuses upon subsequent operations.
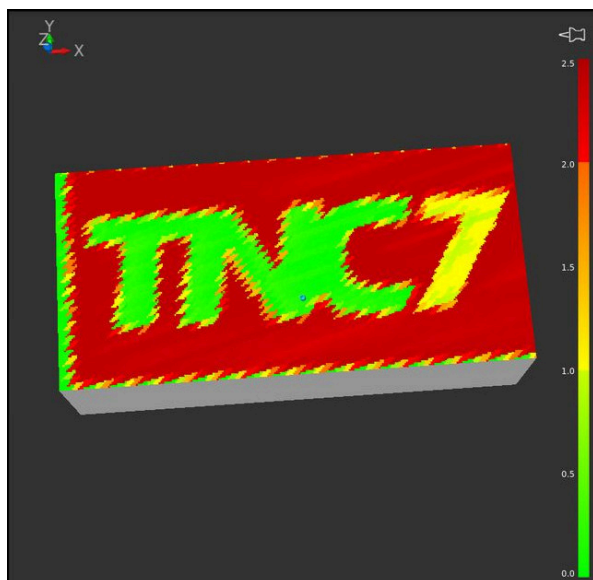
Representation of the component heat map in the simulation with missing pre-machining

Only one component at a time can be monitored with the heatmap. If you start the heatmap several times in a row, monitoring of the previous component is stopped.

Input

11 MONITORING HEATMAP START FOR "Spindle"	; Activate monitoring of the **Spindle** component and display it as a heat map

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Process monitoring 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 Text or string parameter 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.

16.2 Process monitoring (#168 / #5-01-1)

16.2.1 Fundamentals

The control uses process monitoring to detect disturbances in the machining process, e.g.:

- Tool breakage
- Incorrect or missing workpiece pre-machining
- Changed position or size of the workpiece blank
- Wrong material (e.g., aluminum instead of steel)

Process monitoring compares the signal runs of the current execution of an NC program with previous machining operations or with constant values, thereby identifying any possible deviations. In case of deviations, the control reacts with one or several defined reactions. You may, for example, define that the control stops when the spindle current fails due to tool breakage.
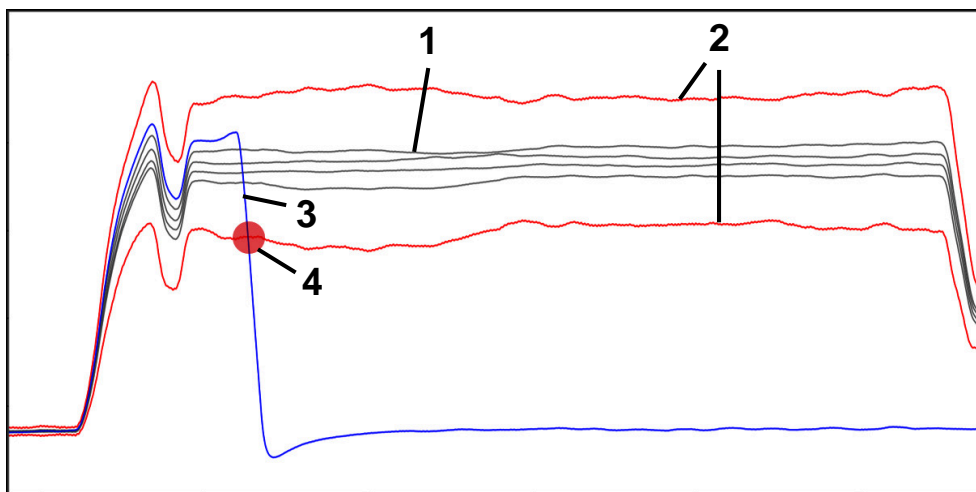
Example: Drop in spindle current due to tool breakage

1	⬛	Recording of machining processes
2	🟥	Limits arising from the recordings and the defined parameters
3	🟦	Current machining operation
4	🔴	A process fault (e.g., due to tool breakage)

> ℹ️ The control cannot monitor every machining process. If the spindle load is too low, the control may not detect a difference from idling (e.g., when finishing with a small oversize).

> ⚙️ Refer to your machine manual.
> This chapter describes the standard functions for process monitoring. The machine manufacturer may change the functions and/or define custom functions.

Definitions

Term	Meaning
Monitoring section	Monitoring sections define the areas in the NC program to be monitored by the control. The monitoring sections contain the **SECTION MONITORING START** and **SECTION MONITORING STOP** syntax elements at the beginning and at the end.
Monitoring task	The control uses the monitoring task to monitor the monitoring sections during the program run.
	A monitoring task consists of a signal, a procedure and one or several reactions.
	The control displays every monitoring task as a graph.
Signal	The signal defines what the control should monitor (e.g., the spindle current).
	The machine provides information about the machining process by means of signals.
Procedure	The procedure defines how the control will monitor the signal (e.g., by waveform comparison).
Reactions	The reactions define how the control reacts in case of a signal deviation (e.g., **Trigger NC stop**).
Parameterization	Parameterization allows adapting the procedure to the machining process if required.
Monitoring template	You can save the parameterization of the current monitoring task, including the defined reactions, as a template. Templates allow you to easily transfer the parameterization to other machines.
	By default, the control will use the templates as monitoring tasks for new monitoring sections or NC programs.
Recordings	The control records the machining operations and displays them in the runtime table. If you assess the recordings, i.e. the results of the machining operation, as "good parts", the control will consider these machining operations as reference machining operations.
Reference machining operation	The control uses the first ten good parts as reference machining operations. For some procedures, the control needs reference machining operations to compare them with the current machining operations (e.g., for waveform comparison).
	Based on the reference machining operations, the control creates a reference signal for monitoring tasks.

> **i** Recordings and settings of prior software versions are not compatible with software version 19. Make sure to delete the old recordings and settings when updating the software. You need to newly set up the monitoring tasks and make new recordings.

16.2.2 First steps in process monitoring

Starting process monitoring

> **i** Use process monitoring only for machining processes with the final feed rate override. Activate process monitoring only after proving-out the workpiece, when the monitored sections of the NC program no longer change.

Start process monitoring as follows:

- ▶ Open the NC program in the **Editor** operating mode
- ▶ Define the start of a monitoring section with **MONITORING SECTION START**
- ▶ Define the end of a monitoring section with **MONITORING SECTION STOP**
- ▶ Select **Select in Program Run**
- > The control opens the NC program in the **Program Run** operating mode.
- ▶ Open the **Process Monitoring** workspace
- ▶ Activate the toggle switch next to the program path
- > The control activates process monitoring for this NC program.
- ▶ Set the feed-rate override to 100 %
- ▶ Press the **NC Start** key
- > Within the monitoring sections, the control shows the recorded signals as graphs.
- > If monitoring tasks with the **Constant** procedure are active, the control will start monitoring at the first machining operation.
- > After program run, the control will stop at the table row of the current machining operation in the runtime table.
- > If no **Part is OK** assessment exists yet, the control will automatically open the **Please evaluate workpiece from last program run** window.
- ▶ If applicable, open the **Form** column
- ▶ Assess the result of the machining operation (e.g., as **Part is OK**)

> **i** The assessments are, for example, required for the **Tunnel** procedure. Depending on the monitoring task, several assessments may be required in order to activate monitoring.

- ▶ Machine further workpieces
- ▶ If applicable, assess the parts resulting from the machining operations in the **Evaluate component** area

> **i** In most cases, you can use the pre-defined monitoring tasks without any edits. If you have to adapt the monitoring tasks to the machining process, you can modify the parametrization of the monitoring tasks.
> **Further information:** User's Manual for Setup and Program Run

Modifying the parametrization of monitoring tasks

To modify the parametrization of monitoring tasks:

- ▶ Activate **Runtime table**, if required

- ▶ Select the desired monitoring section from the right side of the table
- ＞ In the **Form** column, the control displays the monitoring tasks including the recorded machining operations as graphs.
- ▶ If applicable, open the **Form** column that contains the monitoring tasks
- ▶ Open **Settings** within the monitoring task for parameterizing
- ＞ The control shows the selected recording on the left and the preview for the next recording on the right.
- ▶ Adapt the **Parameter settings**, if required
- ▶ Adapt the **Reactions**, if required
- ▶ Select **OK**
- ＞ The control saves the changes and activates them when the NC program is executed the next time.

Changing the monitoring task

To change a monitoring task:

- ▶ Activate **Setup table**, if required
- ＞ The control displays all existing monitoring sections as rows and the six possible monitoring tasks as columns.
- ▶ Select the cell or column to be changed

> **ⓘ** If you select an entire column, you change the monitoring task for all monitoring sections.

- ▶ If applicable, open the **Form** column

- ▶ Select the desired signal in the **Monitoring task** area (e.g., Perpendicular servo lag)
- ▶ Select the desired procedure (e.g., Tunnel)
- ＞ The control only offers the procedures that are permitted for the selected signal.
- ▶ Press **Apply**
- ＞ The control saves your change.
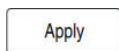
Removing a monitoring task

To remove a monitoring task:

⚙️
- ▶ Activate **Setup table**, if required
- ⟩ The control displays all existing monitoring sections as rows and the six possible monitoring tasks as columns.
- ▶ Select the cell or column to be changed

> ℹ️ If you select an entire column, you remove the monitoring task for all monitoring sections.

📝
- ▶ If applicable, open the **Form** column

[Remove]
- ▶ Select **Remove** in the **Monitoring task** area
- ⟩ The control opens a window with a confirmation prompt.

[OK]
- ▶ Select **OK**
- ⟩ The control removes the monitoring task.

> ℹ️ If you remove a monitoring task, the previous recordings will be kept. You can add the monitoring task again later.

16.2.3 Define monitoring sections with MONITORING SECTION (#168 / #5-01-1)

Application

The NC function **MONITORING SECTION** allows defining monitoring sections for process monitoring in the NC program.

Related topics

- ▪ The **Process Monitoring** workspace

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

Requirement

- ▪ Software option Process Monitoring (#168 / #5-01-1)

Description of function

MONITORING SECTION START is used to define the start of a new monitoring section and **MONITORING SECTION STOP**, to define the end of the monitoring section.

Define a separate monitoring section for each machining step to be monitored. Each monitoring section must be unique. If multiple monitoring sections have the same contents, make sure to name them differently.

HEIDENHAIN recommends ending each monitoring section with **MONITORING SECTION STOP**. Otherwise, the control will end the monitoring section automatically at the end of the program (**END PGM**).

For the following NC functions, the control ends the current monitoring section and starts a new one:

- **MONITORING SECTION START**
- **TOOL CALL** with tool change within a monitoring section
  Further information: "Using TOOL CALL to call a tool", Page 205

The control can only compare the machining operations if the traverses and machining time are identical for each execution. Thus, the monitoring section may only contain the machining operation itself (i.e., it may only begin after the tool call and pre-positioning). The programmed spindle speed must have been reached already.

Note the information on the program structure.

Further information: "Notes on the program structure", Page 507

Input

11 MONITORING SECTION START AS "mill contour"	; Beginning of monitoring section including additional designation

To navigate to this function:

Insert NC function ▶ Special functions ▶ Functions ▶ Process monitoring MONITORING ▶ MONITORING SECTION

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING SECTION	Syntax initiator for the monitoring section of process monitoring
START or **STOP**	Start or end of the monitoring section
AS	Additional designation Optional syntax element Only when **START** is selected

Notes

- The control shows the beginning and the end of the monitoring section in the structure.

  Further information: "The Structure column in the Program workspace", Page 737

- If you change an NC block within a monitoring section, the previous recordings are no longer compatible. Comments are the only items you can change without further impact on monitoring. In order to monitor an edited monitoring section again, delete the existing recordings and define new "good parts".

- If you use different sizes of workpiece blanks, set process monitoring to a more tolerant setting or start the first monitoring section after pre-machining the workpiece blank.

Notes on the program structure

- The following NC functions are prohibited within monitoring sections:
  - Stop of program run (e.g., **M0**, **M1**, or **STOP**)
  - Call of an NC program (e.g., with **CALL PGM**)

    Closed monitoring sections in a called NC program are permitted.

- Some NC functions may cause traverse differences, resulting in deviating machining times. This means that the program sequence is no longer reproducible and thus inadequate for process monitoring.

  Avoid using the following NC functions within monitoring sections:
  - Positions referring to the machine datum (e.g., **M91** or **M92**)
  - Automatic liftoff with **M140 MB MAX**
  - Call of a replacement tool with **M101**
  - Repeats with variable values (e.g., **CALL LBL 99 REP QR1**)
  - Variable jump commands (e.g., **FN 5**)
  - Variable or changing datum shifts (e.g., **TRANS DATUM AXIS XQ1**)
  - Modifications of the spindle speed (e.g., **M3** or **TOOL CALL** with the same tool as before)
  - Combination with AFC sections (e.g., **AFC CUT BEGIN**)

    The AFC function can be used jointly with process monitoring in an NC program. However, the process monitoring sections and AFC sections should not overlap.

- HEIDENHAIN recommends that you program a feed-rate value in the NC block before **MONITORING SECTION START**. Thus, the control will only position the tool once the programmed spindle speed has been reached.

- If you program **MONITORING SECTION STOP** without an associated **MONITORING SECTION START**, the control will display an error.

- The control monitors the movements in the **W-CS** workpiece coordinate system. If you perform the same machining operation at various positions in the machine, make sure to change the workpiece preset and not the workpiece datum.

- When monitoring machining operations with OCM cycles (#167 / #1-02-1), please note the following:
  - Monitor roughing operations only.
  - Always use the same tool (e.g., no resharpening during machining). Small deviations of the tool radius might result in deviating traverses.
  - Pre-position the tool before calling the cycle. If the starting points deviate, OCM will generate different paths.
  - Program the same speed for the cycle as in the tool call.

17

Multiple-axis machining

17.1 Cylinder surface machining with CYLINDER SURFACE (#8 / #1-01-1)

Application

The **CYLINDER SURFACE** NC function allows you to machine the cylinder surface with various NC functions, for example OCM cycles (#167 / #1-02-1), pocket milling cycles or path functions.

Related topics

- Cycles for cylinder surface machining
- OCM cycles
- Pocket milling cycles
- Path functions
  Further information: "Path functions", Page 215

Further information: User's Manual for Machining Cycles

Requirements

- Machine with at least one rotary table axis
  Rotary table axis as modulo axis
- Software option Adv. Function Set 1 (#8 / #1-01-1)
- The cylinder is set up centered and perpendicular on the rotary table
  Workpiece preset in the center and on the surface of the cylinder
- Milling operation **FUNCTION MODE MILL**
- **PARAX COMP DISPLAY** NC function programmed with at least the main axes **X**, **Y** and **Z**
  HEIDENHAIN recommends defining all of the available axes within the **PARAX COMP DISPLAY** function.
- Tool call with tool axis **Z**
- No active coordinate transformation such as **TRANS ROTATION**
- Working plane for cylinder surface machining:
  - Cylinder axis parallel to a machine axis
  - Tool axis parallel to a machine axis and perpendicular above the cylinder axis

> **ⓘ** Machines with axes installed at a right angle or at 45 degrees meet these conditions after tilting the working plane, if required.
> Different kinematics possibly do not allow you to meet these conditions.

Description of function

Use the NC function **CYLINDER SURFACE ON** to activate cylinder surface machining. When the NC function **CYLINDER SURFACE** is active, the control displays an icon in the **Positions** workspace. This icon covers the icon for the **PARAX COMP DISPLAY** NC function.
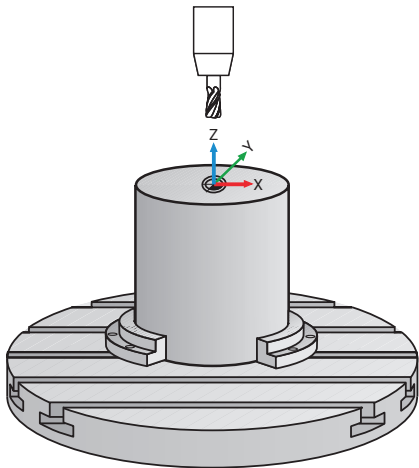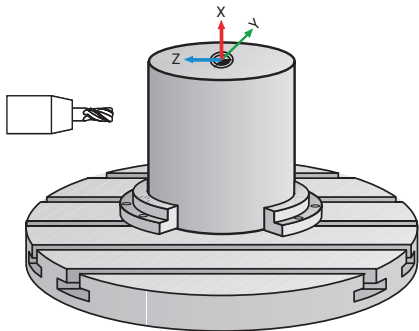
The control deactivates cylinder surface machining in the following cases:

- **CYLINDER SURFACE OFF**
- **M2** or **M30**
- End of program **END PGM**
- Cancellation of an NC program

You program the contour or machining cycles on the unrolled surface of the cylinder. The control transfers the programmed values to the cylinder surface. The control automatically calculates the feed rate of the rotary table axis based on the programmed feed rate and the cylinder diameter.

Use the **X** and **Y** coordinates to program the contour or machining cycles, independent of which rotary axes exist on your machine. The X coordinate describes the circumference of the cylinder and defines the position of the rotary table axis. The Y coordinate is on the cylinder axis. The Z axis serves as infeed axis.

The following table shows a possible sequence for cylinder surface machining:

Description	Help graphic
The workpiece preset is in the center and on the surface of the cylinder.	
You tilt the working plane to the spatial angle **SPB-90** and position the tool in the Y axis on the value 0. The working plane is tilted to the spatial angle **SPB-90**. The tool is thus oriented perpendicularly above the cylinder axis. Due to the tilted working plane, the cylinder axis and the tool axis are each parallel to a machine axis.	

Description	**Help graphic**
You activate the NC function **CYLINDER SURFACE**. The control automatically shifts the workpiece datum in the direction of the tool axis on the cylinder surface: ■ The X coordinate describes the circumference of the cylinder and defines the position of the rotary table axis ■ The Y coordinate is on the cylinder axis ■ The Z axis serves as infeed axis	
You shift the workpiece datum in the direction Y-.	
You program the contour on the unrolled surface of the cylinder.	
The completed contour is transferred to the cylinder surface.	
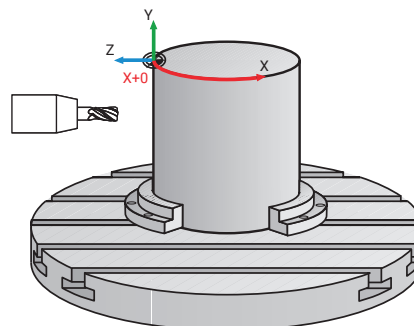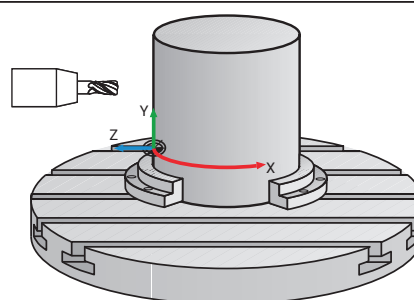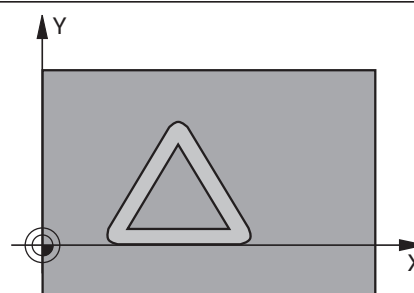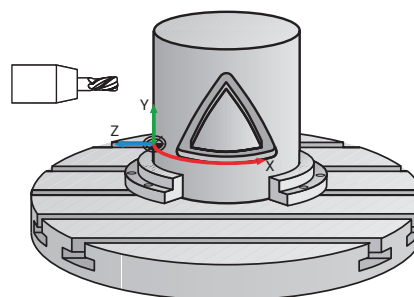
🛈   If the **CYLINDER SURFACE** NC function is active, the tool is positioned perpendicularly to the cylinder surface and as a result, the tool center is aligned with the cylinder center. If the X coordinate changes, the control moves the rotary table axis and not the tool.

This results in the following effects:

■ When using a contour definition with Y coordinates, the walls are not parallel to each other.

■ The bottom of a pocket, for example, can be uneven.

■ When you produce threads using thread milling cycles, the threads will be conical.

    Only use tapping cycles for cylinder surface machining.

    Further information: User's Manual for Machining Cycles

If cylinder surface machining is active, do not use the following NC functions:

- **M91/M92**
- **TOOL CALL**
- **M140**
- **M144** (#9 / #4-01-1)
- **POLARKIN**
- Facing head with **FACING HEAD POS** (#50 / #4-03-1)
- Tool radius compensation
- 3D tool compensation (#9 / #4-01-1)
- 3D tool radius compensation depending on the contact angle (#92 / #2-02-1)
- **FUNCTION SHAPING** (#96 / #7-04-1)
- **FUNCTION TCPM** or **M128** (#9 / #4-01-1)
- Rotary axis movements
- Tilting the working plane with **PLANE** functions
- Switching the machining mode with **FUNCTION MODE**
- Handwheel superimpositioning with **M118**
- Handwheel superimpositioning with the global program settings GPS (#44 / #1-06-1)

Input

CYLINDER SURFACE ON

11 CYLINDER SURFACE ON D99 X AS LIN	; Activate cylinder surface machining and define the cylinder size

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Cylinder kinematics ▶ CYLINDER SURFACE ON

The NC function includes the following syntax elements:

Syntax element	Meaning
CYLINDER SURFACE ON	Syntax initiator for activating cylinder surface machining
R or **D**	Radius or diameter of the cylinder Number or numerical parameter
X AS	Axis of the unrolled surface of the cylinder
LIN or **DEG**	Indication of coordinates defining the unrolled surface of the cylinder as length or angle **DEG** currently has no function If **DEG** is selected, the control will display the error message **Block format incorrect**.

CYLINDER SURFACE OFF

| **11 CYLINDER SURFACE OFF** | ; Deactivate cylinder surface machining |

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Cylinder kinematics ▶ CYLINDER SURFACE OFF

The NC function includes the following syntax elements:

Syntax element	Meaning
CYLINDER SURFACE OFF	Syntax initiator for deactivating cylinder surface machining

Note

If a basic rotation around the cylinder axis is active, you always must tilt the working plane using, for example, **PLANE SPATIAL SPA+0 SPB+0 SPC+0** before machining the cylinder surface.

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

17.1.1 Program structure for cylinder surface machining

Here you see a possible program structure for cylinder surface machining.

	BLK FORM...	
	TOOL CALL...	
If required, tilt the working plane	**PLANE SPATIAL...**	
Pre-position above the cylinder axis	**L X... Y+0 Z...**	
Activate cylinder surface machining	**CYLINDER SURFACE ON...**	
Shift datum, if required	**TRANS DATUM...**	
Machine cylinder surface	**CYCL DEF 251 RECTANGULAR POCKET**	; E.g., pocket milling cycle
	CYCL CALL...	
Reset datum shift	**TRANS RESET**	
Deactivate cylinder surface machining	**CYLINDER SURFACE OFF**	
If required, reset tilt angle and deactivate tilting of the working plane	**PLANE RESET...**	
...		

Definition

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.

17.2 Working with the parallel axes U, V and W

17.2.1 Fundamentals

In addition to the main axes X, Y, and Z, the parallel axes U, V, and W, are available. A parallel axis is, for example, a spindle sleeve for boring so that smaller masses are moved on large machines.

Further information: "Programmable axes", Page 124

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 515

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

  Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 520

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.

17.2.2 Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP

Application

The **FUNCTION PARAXCOMP** function is used to define whether the control takes parallel axes into account in the traversing movements with the associated main axis.

Description of function

If the **FUNCTION PARAXCOMP** function is active, the control displays an icon in the **Positions** workspace. The icon for **FUNCTION PARAXMODE** may cover an active icon for **FUNCTION PARAXCOMP**.

Further information: 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 518

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 515

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 126

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:	Position-display values:
■ **RFACTL** mode: **Z+0**, **W-30**	■ **RFACTL** mode: **Z-10**, **W-30**
■ **ACTL.** mode: **Z-30**, **W-20**	■ **ACTL.** mode: **Z-30**, **W-20**
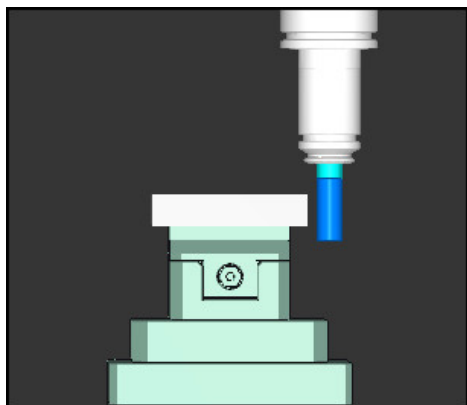
The tool tip is lower by the offset value than programmed in the NC program (**RFACTL W-30** instead of **W-20**).
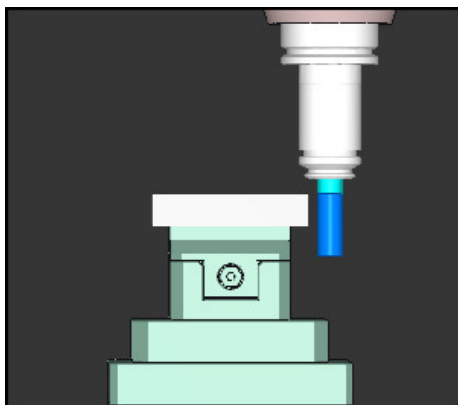
The tool tip is lower by the twice the offset value than programmed in the NC program (**RFACTL Z-10**, **W-30** instead of **Z+0**, **W-20**).

> **ℹ** If you only move the W axis while the **PARAXCOMP DISPLAY** function is active, the control takes the offset into account only once, independent of the setting of the **presetToAlignAxis** machine parameter.

17.2.3 Select three linear axes for machining with FUNCTION PARAXMODE

Application

Use the **PARAXMODE** function to define the axes the control is to use for machining. You program all traverses and contour descriptions in the main axes X, Y and Z, independent of your machine.

Requirement

- Parallel axis is included in the calculation

    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 515

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 521

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 **END PGM**
- **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 224

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.

17.2.4 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

You cannot use the following cycles with parallel axes:

■ Cycle **285 DEFINE GEAR** (#157 / #4-05-1)

■ Cycle **286 GEAR HOBBING** (#157 / #4-05-1)

■ Cycle **287 GEAR SKIVING** (#157 / #4-05-1)

■ Touch-probe cycles

17.2.5   Example

Drilling is carried out with the W axis in the following NC program:

0 BEGIN PGM PAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S2222	; Call the tool in the tool axis **Z**
4 L Z+100 R0 FMAX M3	; Position the main axis
5 CYCL DEF 200 DRILLING	
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 FUNCTION PARAXCOMP DISPLAY Z	; Activate display compensation
7 FUNCTION PARAXMODE X Y W	; Positive axis selection
8 L X+50 Y+50 R0 FMAX M99	; The parallel axis **W** executes the infeed
9 FUNCTION PARAXMODE OFF	; Restore the standard configuration
10 L M30	
11 END PGM PAR MM	

17.3 Using a facing head with FACING HEAD POS (#50 / #4-03-1)

Application

A facing head, also called facing slide, allows you to perform almost all turning operations with fewer different tools. The position of the facing head is programmable in the X direction. On the facing head, you mount, for example, a longitudinal turning tool that you call with a TOOL CALL block.

Related topics

■ Machining with parallel axes **U**, **V** and **W**

Further information: "Working with the parallel axes U, V and W", Page 515

Requirements

■ Software option Turning (#50 / #4-03-1)

■ Control prepared by the machine manufacturer

The machine manufacturer must take the facing head into account in the kinematics.

■ Kinematics with facing head activated

Further information: "Switching the operating mode with FUNCTION MODE", Page 154

■ Workpiece datum in the working plane is at the center of the rotationally symmetrical contour

With a facing head, the workpiece datum must not be in the center of the rotary table, because the tool spindle rotates.

Further information: "Datum shift with TRANS DATUM", Page 334

Description of function

> ⚙ Refer to your machine manual.
> The machine manufacturer can provide customized cycles for working with a facing head. The standard functionality is described below.

The facing head is defined as a turning tool.

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

Please note for tool calls:

■ **TOOL CALL** block without tool axis

■ Cutting speed and spindle speed with **TURNDATA SPIN**

■ Switch the spindle on with **M3** or **M4**

Machining also works with a tilted working plane and on workpieces that are not rotationally symmetric.

If you move with the facing head without the **FACING HEAD POS** function, you must program the motions of the facing head with the U axis (e.g., in the **Manual operation** application). If the **FACING HEAD POS** function is active, program the facing head with the X axis.

When you activate the facing head, the control automatically positions itself at the workpiece datum in **X** and **Y**. To avoid collisions, you can define a safe height using the **HEIGHT** syntax element.

The facing head is deactivated with the **FUNCTION FACING HEAD** function.

Input

Activating the facing head

11 FACING HEAD POS HEIGHT+100 FMAX	; Activate facing head and move with rapid traverse to safe height **Z**+100

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Turning functions ▶ Facing slide ▶ FACING HEAD POS

The NC function includes the following syntax elements:

Syntax element	Meaning
FACING HEAD POS	Activate the syntax initiator for the facing head
HEIGHT	Safe height in the tool axis Optional syntax element
F or **FMAX**	Approach safe height with defined feed rate or rapid traverse Optional syntax element
M	Additional function Optional syntax element

Deactivating the facing head

11 FUNCTION FACING HEAD OFF	; Deactivate facing head

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Turning functions ▶ Facing slide ▶ FUNCTION FACING HEAD OFF

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FACING HEAD OFF	Deactivate the syntax initiator for the facing head

Notes

NOTICE

Caution: Danger to the tool and workpiece!

For a facing head to be used, a kinematics model prepared by the machine manufacturer must be selected by means of the **FUNCTION MODE TURN** function. In this kinematics model, the control implements the programmed X-axis movements of the facing head as U-axis movements when the **FACING HEAD** function is active. When the **FACING HEAD** function is not active and in **Manual operation** operating mode, this automated implementation does not take place. As a result, **X** axis movements (programmed or via axis key) will be performed in the X axis. In this case, the facing head has to be moved with the U axis. There is a danger of collision during retraction or manual movements!

▶ Position the facing head at its home position while the **FACING HEAD POS** function is active

▶ Retract the facing head while the **FACING HEAD POS** function is active

▶ In the **Manual operation** operating mode, move the facing head with the **U** axis key.

▶ As the **Tilt working plane** function can be used, pay attention to the 3D ROT status

---

- To set a spindle-speed limitation, you can use the **NMAX** value from the tool table as well as the **SMAX** value from **FUNCTION TURNDATA SPIN**.
- The following constraints apply to the use of a facing head:
  - Miscellaneous functions **M91** and **M92** cannot be used
  - Retraction with **M140** is not possible
  - **TCPM** or **M128** are not possible (#9 / #4-01-1)
  - **DCM** collision monitoring cannot be used (#40 / #5-03-1)
  - Cycles **800**, **801**, and **880** cannot be used
  - Cycles **286** and **287** cannot be used (#157 / #4-05-1)
- If you are using the facing head in the tilted working plane, please note the following:
  - The control calculates the tilted working plane as in milling mode. The **COORD ROT** and **TABLE ROT** functions, as well as **SYM (SEQ)**, reference the XY plane.
    Further information: "Tilting solution", Page 384
  - HEIDENHAIN recommends selecting the **TURN** positioning behavior. The **MOVE** positioning behavior is not the best option in combination with the facing head.
    Further information: "Rotary axis positioning", Page 381

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. If **FACING HEAD POS** is used, the machine parameter applies
to the parallel axis (**U** axis) only (**U_OFFS**).

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

- If the machine parameter has not been defined or has been set to **FALSE**, the
  control does not take the offset into account during machining.
- If the machine parameter axis has been set to **TRUE**, the offset can be used to
  compensate for a facing head offset. If you are using a facing head with multiple
  tool clamp options, set the offset for the current clamping position. This ensures
  that you can run NC programs independent of the tool clamping position.

17.4 Workpiece shaping with FUNCTION SHAPING (#96 / #7-04-1)

Application

Contour planing, also known as shaping, enables you to create sealing surfaces with a high surface definition, for example. When **FUNCTION SHAPING** is active, the control automatically moves the tool toward the contour during traverse movements. Using **FUNCTION SHAPING**, this automatic tracking also enables you to perform engraving, engine turning, or beveling.

Contour planing is performed with turning tools (e.g., recess-turning tools **RECTURN**)

Related topics

- Entering tool data in the tool management

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

- Compensating turning tools in the NC program with **FUNCTION TURNDATA CORR**

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR", Page 422

Requirements

- Kinematics description

  In order to track the tool, the control requires a kinematics description prepared by the machine manufacturer.

- Software option Adv. Spindle Interpol. (#96 / #7-04-1)
- Tool definition
  - Tool type **Turning tool**
  - Tool tip direction with **DIRECT**
  - Tool axis **Z**
- **FUNCTION MODE MILL** active

Description of function

For shaping, activate a machine kinematics model in which the tool spindle is defined as rotary axis. This way, the control can track the tool along the contour.

You can use a basic rotation or 3D basic rotation to align the workpiece and carry out shaping even with a tilted working plane.

The control resets **FUNCTION SHAPING** in the following cases:

- **FUNCTION SHAPING END**
- **M30**
- **Internal stop**

Contour definition

Program the contour along which the control tracks the tool in **FUNCTION SHAPING**.

During shaping, the tool must be positioned perpendicular to the working plane. If you program the rotary axes within **FUNCTION SHAPING**, the control will display an error message.

Program the contour using only the following NC functions:

- Path functions except for approach and departure functions, without tool radius compensation
- **TRANS DATUM**
- **TRANS ROT** or Cycle **10 ROTATION**

> ℹ️ HEIDENHAIN recommends programming only the contour to be shaped within the **FUNCTION SHAPING** function. For example, if you are performing pre-positioning for the next contour, program the traverse movements after **FUNCTION SHAPING END**.

Tools for shaping

The required NC functions and software options for shaping vary, depending on the turning tool being used.

If the tool cutting edge is in the spindle center, you do not need the Adv. Function Set 2 software option (#9 / #4-01-1) for shaping. If the tool cutting edge is outside the spindle center, the Adv. Function Set 2 software option (#9 / #4-01-1) is required for shaping.

Tool cutting edge in spindle center
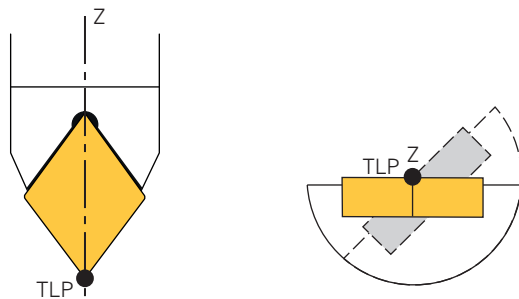
Tool location point TLP in spindle center, front view and view from below

If the **XL** and **YL** parameters of the tool include the value 0, the tool location point TLP is in the spindle center. If you program **FUNCTION TURNDATA CORR-TCS**, the **DXL** syntax element must also include the value 0.

During shaping, the control turns the spindle in order to track the tool along the contour. While the spindle is turning, the tool location point remains at the same place for these tools.

Tool cutting edge outside the spindle center

Tool location point TLP outside spindle center, front view and view from below

If the **XL** or **YL** parameters of the tool include a value unequal to 0, the tool location point TLP is outside the spindle center. If you program the **DXL** syntax element to be unequal to 0 inside **FUNCTION TURNDATA CORR-TCS**, you will also shift the tool location point.
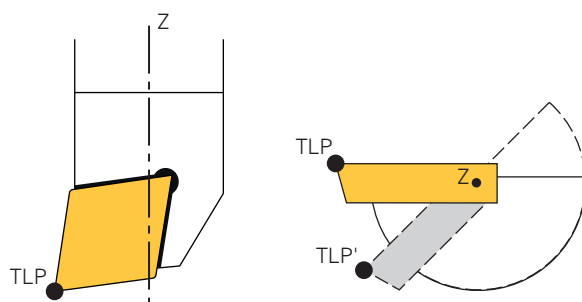
During shaping, the control turns the spindle in order to track the tool along the contour. Rotation of the spindle leads to an offset regarding the original position of the cutting edge on these tools. The view from below shows you the offset at the tool location point TLP'. Without compensation, the tool would move away from the contour or damage the contour.

To compensate for this offset during machining and to keep the tool permanently at the contour, program **M128** or **FUNCTION TCPM** with the selection of **AXIS POS** (#9 / #4-01-1).

Further information: "Tool location point (TLP, tool location point)", Page 202

Input

FUNCTION SHAPING BEGIN

11 FUNCTION SHAPING BEGIN	; Activate shaping

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Shaping SHAPING ▶ FUNCTION SHAPING BEGIN

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION SHAPING BEGIN	Syntax initiator for activating tracking

FUNCTION SHAPING END

11 FUNCTION SHAPING END	; Deactivate shaping

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ Shaping SHAPING ▶ FUNCTION SHAPING END

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION SHAPING END	Syntax initiator for deactivating tracking

Notes

> ### NOTICE
>
> **Caution: Danger to the tool and workpiece!**
>
> If **FUNCTION SHAPING** is active, the control tracks the tool along the contour. If the contour includes small inner radii, then large tool carriers or tools that have not been relief-ground can damage the contour during tracking.
>
> ▶ For shaping tools, observe the information of the tool manufacturer about the minimum permissible inner radius
> ▶ Use relief-ground tools with suitable tool carriers

■ If **FUNCTION SHAPING** is active, the control calculates the bisector from the current NC block and the next NC block. During the traverse movement, the control turns the spindle and thus the tool cutting edge. At the end of each NC block the tool cutting edge is positioned on the bisector to the contour.

■ Refer to your machine manual.

  If the shaping kinematics model is active, the tool spindle acts as an additional rotary axis. The tilting functions of the control permit only two rotary axes. To program a tilting function with the shaping kinematics model active, you must exclude the tool spindle from the calculation using **M138**.

  In connection with the shaping kinematics model, the control activates **M138** on a standard basis, if applicable.

■ The NC function **FUNCTION TURNDATA CORR** is included in the scope of functionality of the Adv. Spindle Interpol. software option (#96 / #7-04-1).

  You can use the NC function **TURNDATA CORR-TCS** just like a tool radius compensation to program the contour with the drawing dimensions.

17.4.1 Program structure for workpiece shaping

Here you see a possible program structure for workpiece shaping. Optional steps begin with **If required**. The third column contains further information or conditions for optional steps.

	BLK FORM...	
Call tool for workpiece shaping	**TOOL CALL ...**	; Turning tool with tool axis **Z** required
If required, compensate for the tool	**FUNCTION TURNDATA CORR-TCS...**	
Activate shaping kinematics	**FUNCTION MODE MILL "..."**	
If required, exclude spindle from tilting using M138	**M138...**	; Only if you tilt the working plane after the shaping kinematics is active
If required, tilt the working plane	**PLANE SPATIAL...**	
If required, activate FUNCTION TCPM using AXIS POS	**FUNCTION TCPM ... AXIS POS ...**	; Only if the tool cutting edge is outside the spindle center
Activate FUNCTION SHAPING	**FUNCTION SHAPING BEGIN**	; Activate tracking
Shaping	**L X... Y... Z...**	; Only linear axis movements permitted
	CC ...	
	C X... Y...	
Deactivate FUNCTION SHAPING	**FUNCTION SHAPING END**	
If required, deactivate FUNCTION TCPM	**FUNCTION RESET TCPM**	
If required, reset the tilted working plane	**PLANE RESET...**	
Activate milling kinematics	**FUNCTION MODE MILL "..."**	
...		

Definitions

Engine turning
Engine turning is a process of mechanical engraving to produce patterns made of overlapping lines. This technique is used, for example, in printing technology and in the watchmaking and jewelery industries.

Beveling
Beveling is a special technique used to produce edges of highest surface quality, for example, in the watchmaking and jewelery industries.

17.5 Machining with polar kinematics with POLARKIN

Application

In a polar kinematic model, the path contours of the working plane are performed by one linear axis and one rotary axis instead of by two linear principal axes. The working plane is defined by the linear principal axis and the rotary axis while the working space is defined by these two axes and the infeed axis.

On milling machines, various linear principal axes can be replaced with suitable rotary axes. For example on large machines, polar kinematics enable you to machine much larger surfaces than with only the principal axes.

On turning and grinding machines that have only two linear principal axes, polar kinematics enable milling operations to be performed on the front face.

Requirements

■ Machine with at least one rotary axis

The polar rotary axis must be installed onto the table side so that it is opposite the selected linear axes and must be configured as a modulo axis. Thus, the linear axes must not be positioned between the rotary axis and the table. The maximum range of traverse of the rotary axis is limited by the software limit switches if necessary.
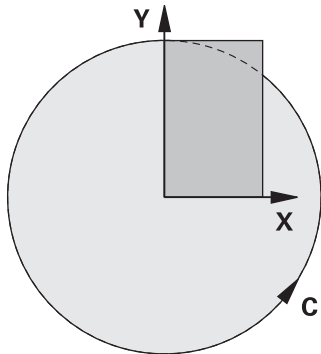
■ **PARAX COMP DISPLAY** NC function programmed with at least the main axes **X**, **Y** and **Z**

HEIDENHAIN recommends defining all of the available axes within the **PARAX COMP DISPLAY** function.

Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 515

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.

The function **POLARKIN AXES** allows you to activate the polar kinematics model with two linear axes and one rotary axis.

- 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.
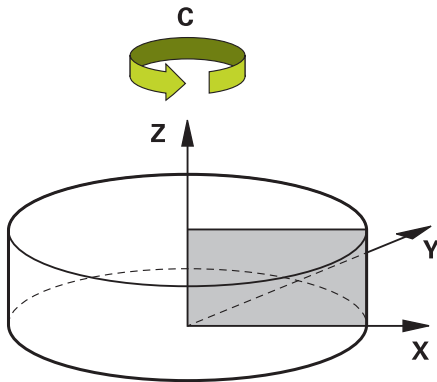
The following scenarios lead to deactivation of the polar kinematics:

- 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 pole is the rotation center of the rotary axis which you define within **POLARKIN**.

The control provides the following options for machining at the pole:

POLE options:

Syntax	Function
ALLOWED	The control permits machining operations at the pole
SKIPPED	The control prevents machining operations at the pole

> **ℹ** The disabled area corresponds to a circular surface with a radius of 0.001 mm (1 µm) around the pole.

Input

11 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
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 534
	Only when **AXES** is selected
POLE:	Selection of machining in the pole
	Further information: "POLE options", Page 534
	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**.
- The workpiece preset does not need to be in the pole.
- 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 558
  - 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 548
  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 **preset-ToAlignAxis** (no. 300203) to define for each axis how the control will interpret offset values. For **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 for 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 316

  - If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate for a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

17.5.1 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 PARAX COMP DISPLAY X Y Z	; Activate **FUNCTION PARAX COMP 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**
7 * - ...	; Datum shift in polar kinematics
8 TRANS DATUM AXIS X+50 Y+50 Z+0	
9 CYCL DEF 14.0 CONTOUR	
10 CYCL DEF 14.1 CONTOUR LABEL2	
11 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	
12 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	
13 M99	
14 TRANS DATUM AXIS X+0 Y+0 Z+0	
15 POLARKIN OFF	; Deactivate **POLARKIN**
16 FUNCTION PARAX COMP OFF X Y Z	; Deactivate **FUNCTION PARAX COMP DISPLAY**
17 L X+0 Y+0 Z+10 A+0 C+0 FMAX	
18 M30	
19 LBL 2	
20 L X-20 Y-20 RR	
21 L X+0 Y+20	
22 L X+20 Y-20	
23 L X-20 Y-20	
24 LBL 0	
25 END PGM POLARKIN_SL MM	

17.6 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 5-axis simultaneous machining of free-form surfaces.
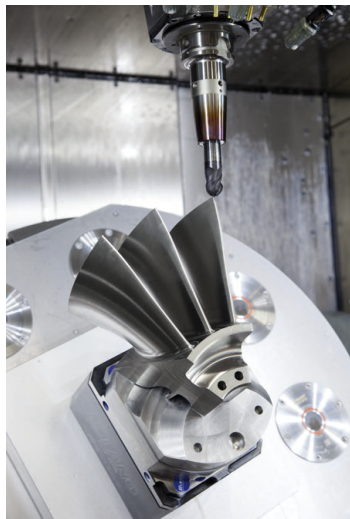

For CAM-generated NC programs to be able to use the full performance potential of the control and to provide you with such options as intervention and correction, certain requirements must be met.

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 543

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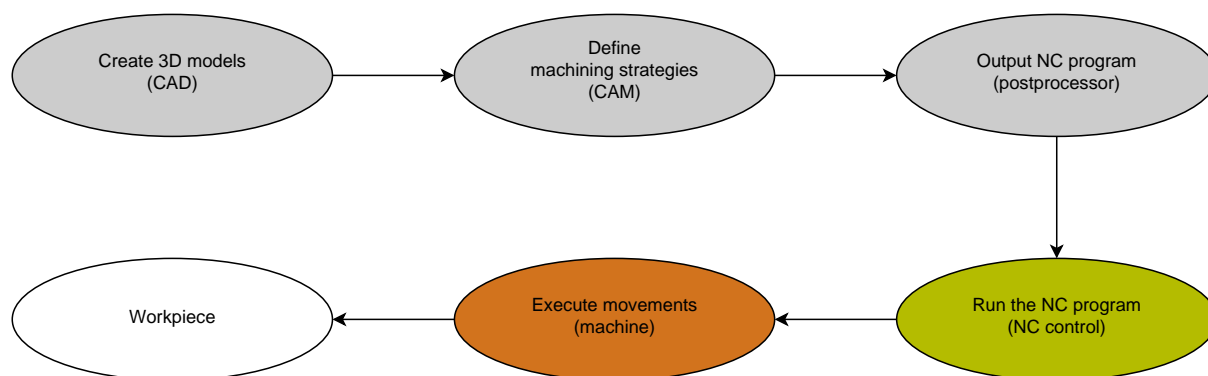
Related topics

■ Using 3D data directly at the control

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

■ Programming graphically

Further information: "The Contour graphics workspace ", Page 675

17.6.1 Output formats of NC programs

Output in HEIDENHAIN Klartext format

If you output the NC program in Klartext, you have the following options:

- 3-axis output
- Output with up to five axes, without **M128** or **FUNCTION TCPM**
- Output with up to five axes, with **M128** or **FUNCTION TCPM** (#9 / #4-01-1)

> **i** Prerequisites for 5-axis-machining:
> - Machine with rotary axes
> - Software option Adv. Function Set 1 (#8 / #1-01-1)
> - Software option Adv. Function Set 2 (#9 / #4-01-1) for **M128** or **FUNCTION TCPM**

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.
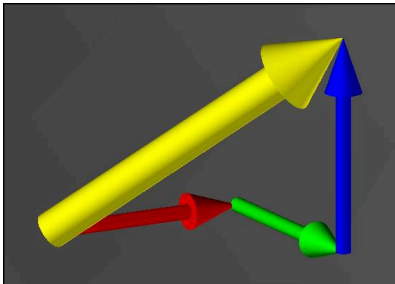
Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

Further information: "Presets on the tool", Page 199

Examples

11 L X+88 Y+23.5375 Z-8.3 R0 F5000	; 3-axis

11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000	; 5-axis without **M128**

11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000 M128	; 5-axis with **M128**

Output with vectors

From the point of view of physics and geometry, a vector is a directed variable that describes a direction and a length.

When outputting with vectors, the control requires at least one vector that specifies the direction of the surface normal or the tool angle of inclination. Optionally, the NC block contains both vectors.

> **i** Prerequisites:
> - Machine with rotary axes
> - Software option Adv. Function Set 1 (#8 / #1-01-1)
> - Software option Adv. Function Set 2 (#9 / #4-01-1)

> **i** You can only use the output with vectors in milling mode.
> **Further information:** "Switching the operating mode with FUNCTION MODE", Page 154

> **i** Vector output with the direction of the surface normals is required for using 3D tool radius compensation depending on the tool angle (#92 / #2-02-1).
> **Further information:** "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439

Examples

11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258	; 3-axis with surface normal vector, without tool orientation

11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258 TX+0.0078922 TY-0.8764339 TZ+0.2590319 M128	; 5-axis with M128, surface normal vector and tool orientation

Structure of an NC block with vectors
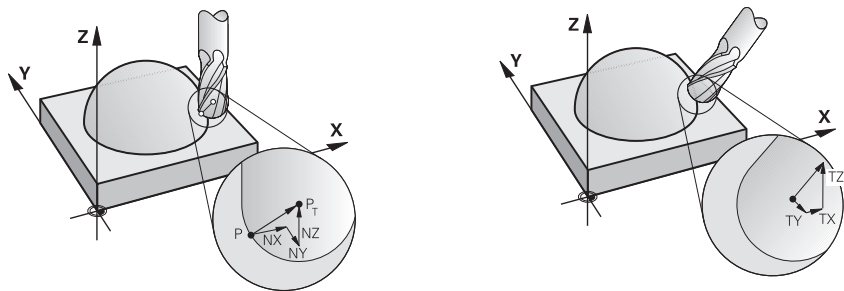
Surface normal vector perpendicular to the contour


Tool direction vector

Example

11 LN X+0.499 Y-3.112 Z-17.105 NX0 NY0 NZ1 TX+0.0078922 TY-0.8764339 TZ+0.2590319	; Straight line **LN** with surface normal vector and tool orientation

Syntax element	Meaning
LN	Straight line **LN** with surface normal vector
X Y Z	Target coordinates
NX NY NZ	Components of the surface normal vector
	Optional syntax element
TX TY TZ	Components of the tool direction vector
	Optional syntax element

17.6.2 Types of machining according to number of axes

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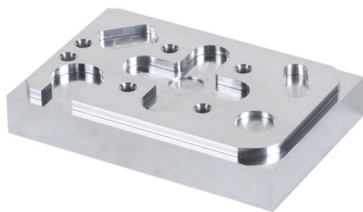
If only the linear axes **X**, **Y** and **Z** are required for machining a workpiece, 3-axis machining takes place.

3+2-axis machining

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

> **i** Prerequisites:
> - Machine with rotary axes
> - Software option Adv. Function 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.

> **i** Prerequisites:
> - Machine with rotary axes
> - Software option Adv. Function Set 1 (#8 / #1-01-1)
> - Software option Adv. Function Set 2 (#9 / #4-01-1)

5-axis machining

In 5-axis machining, also referred to as 5-axis simultaneous machining, the machine moves five axes at the same time. For free-form surfaces, this means that the tool can always be oriented perfectly with respect to the workpiece surface.

> **i** Prerequisites:
> - Machine with rotary axes
> - Software option Adv. Function Set 1 (#8 / #1-01-1)
> - Software option Adv. Function Set 2 (#9 / #4-01-1)
>
> 5-axis machining is not possible with the export version of the control.

17.6.3 Process steps

CAD

Application

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

Notes

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

> **i** Support manufacturing with additional files:
> - Provide 3D models in STL format. The control-internal simulation can use the CAD data as blank and finished parts, for example. Additional models of tool and workholding equipment are required in conjunction with collision testing (#40 / #5-03-1).
> - Provide drawings with the dimensions to be checked. The file type of the drawings is not important in this respect, since the control can also open files such as PDFs, and therefore supports paperless production.

Definition

| Abbreviation	Definition
CAD (computer-aided design)	Computer-aided design

CAM and postprocessor

Application

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

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

Notes on CAD data

- Avoid quality losses due to unsuitable transfer formats. Integrated CAM systems with manufacturer-specific interfaces work in some cases without loss.
- Take advantage of the available accuracy of the CAD data obtained. A geometry or model error of less than 1 µm is recommended for finishing large radii.

Notes on chord errors and Cycle 32 TOLERANCE
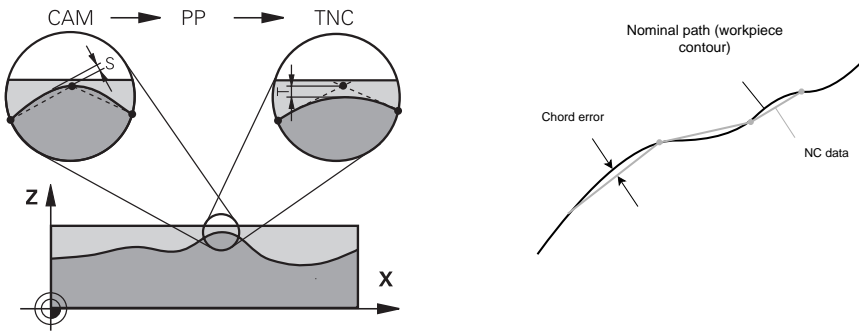
- In roughing, the focus is on the processing speed.
  The sum of the chord error and the tolerance **T** in Cycle **32 TOLERANCE** must be smaller than the contour allowance, otherwise contour violations may occur.

Chord error in CAM system	0.004 mm to 0.015 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.05 mm to 0.3 mm

- When finishing with the aim of high accuracy, the values must provide the required data density.

|	
Chord error in CAM system	0.001 mm to 0.004 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.002 mm to 0.006 mm

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

Chord error in CAM system	0.001 mm to 0.005 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.010 mm to 0.020 mm

Further information: 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.
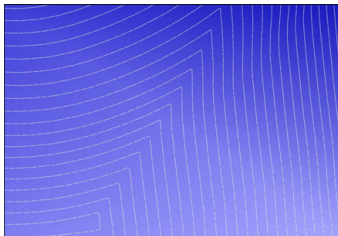
Tool paths with strong changes in direction at transitions

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 348

- 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 393

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

Further information: "Notes on software limit switches for modulo axes", Page 548

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 199

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, QS parameters and named parameters", Page 599

■ 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 281

■ 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 736

■ Use comments to communicate important information about the NC program such as the chord error being used.

Further information: "Adding comments", Page 734

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 761

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

Further information: "Notes on software limit switches for modulo axes", Page 548

■ 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 539

Further information: "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439

■ Software options enable further optimizations.

Further information: "Functions and function packages", Page 549

Further information: "Software options", Page 63

Notes on software limit switches for modulo axes

> **i** The following information on software limit switches for modulo axes also applies to traversing limits.
> **Further information:** 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°.

17.6.4 Functions and function packages

ADP motion control
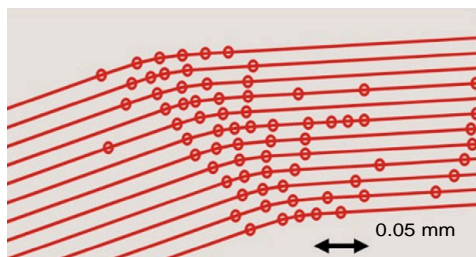
Distribution of points
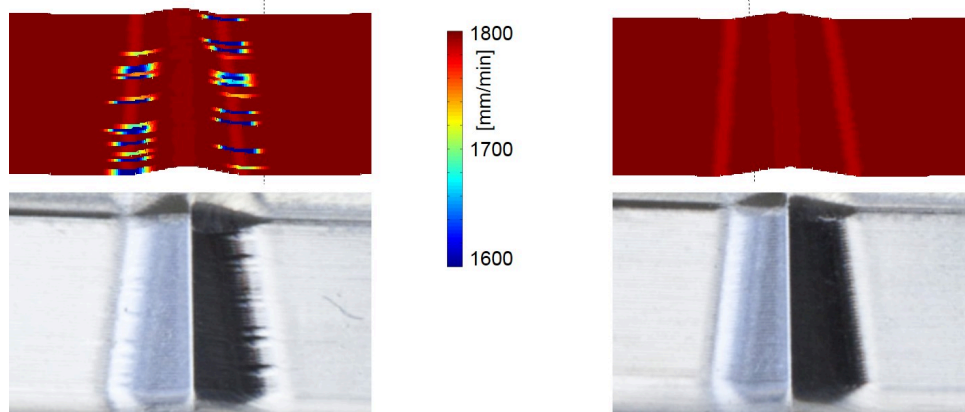
Comparison without and with ADP

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
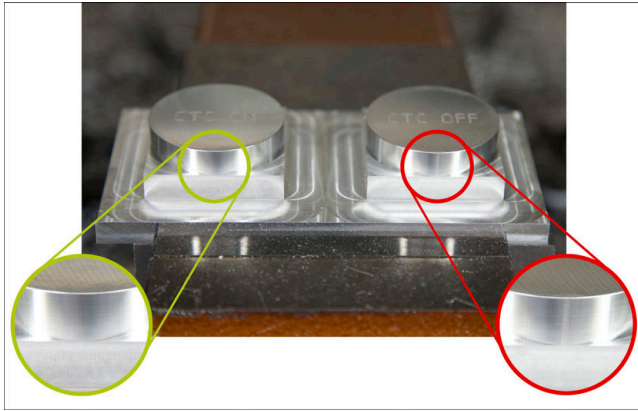
The Dynamic Precision package of functions enables you to machine quickly and accurately, and with high surface quality.

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.**

18

Miscellaneous
Functions

18.1 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.

18.1.1 Programming the STOP function

To program the **STOP** function:

| STOP |

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

Note

Refer to your machine manual.

In turning mode, miscellaneous functions for the turning spindle must be programmed using different numbers (e.g., **M303** instead of **M3** (#50 / #4-03-1)). The machine manufacturer defines the numbers to be used.

Using the optional machine parameter **CfgSpindleDisplay** (no. 139700), the machine manufacturer defines the miscellaneous function numbers to be displayed in the status display.

18.2 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 558
M92 Traverse in the **M92** coordinate system	□	Page 560
M94 Reduce the display for rotary axes to under 360°	□	Page 562
M97 Machine small contour steps	■	Page 563
M98 Machine open contours completely	■	Page 565
M99 Call a cycle once per block	■	See the User's Manual for Machining Cycles
M101 Automatically insert a replacement tool	□	Page 591
M102 Reset **M101**	■	
M103 Reduce feed rate for infeed movements	□	Page 566
M107 Permit positive tool oversizes	□	Page 593
M108 Check the radius of the replacement tool Reset **M107**	■	Page 595
M109 Adapt feed rate for circular paths	□	Page 567
M110 Reduce feed rate for inner radii	□	
M111 Reset **M109** and **M110**	■	
M116 Interpret feed rate for rotary axes as mm/min	□	Page 569
M117 Reset **M116**	■	
M118 Activate handwheel superimpositioning	□	Page 570
M120 Pre-calculate the radius-compensated contour (look ahead)	□	Page 571
M126 Shorter-path traverse of rotary axes	□	Page 575
M127 Reset **M126**	■	

Function	Effect	Further information
M128 Automatically compensate for tool inclination (TCPM)	□	Page 576
M129 Reset **M128**	■	
M130 Traverse in the non-tilted input coordinate system **I-CS**	□	Page 561
M136 Interpret feed rate as mm/rev	□	Page 581
M137 Reset **M136**	■	
M138 Take rotary axes into account during machining operations	□	Page 582
M140 Retract in the tool axis	□	Page 584
M141 Suppress touch probe monitoring	□	Page 596
M143 Rescind basic rotations	□	Page 586
M144 Factor the tool offset into the calculations	□	Page 586
M145 Reset **M144**	■	
M148 Automatically lift off upon an NC stop or a power failure	□	Page 588
M149 Reset **M148**	■	
M197 Prevent rounding off of outside corners	■	Page 589

18.3 Miscellaneous functions for coordinate entries

18.3.1 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 312

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 126

> ⚙ 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 126
- 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 764
- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

18.3.2 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 126

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 126

> ⚙ 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 126
- 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 764
- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

18.3.3 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 224

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 321

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.

18.4 Miscellaneous functions for path behavior

18.4.1 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 384

- 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°.

18.4.2 Machining small contour steps with M97

Application

With **M97** you can produce contour steps that are smaller than the tool radius. The control does not damage the contour and does not issue an error message.

> **i** HEIDENHAIN recommends using the more powerful function **M120** 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**

  Further information: "Pre-calculating a radius-compensated contour with M120", Page 571

Description of function

Effect

M97 is in effect blockwise and takes effect at the end of the block.

Application example
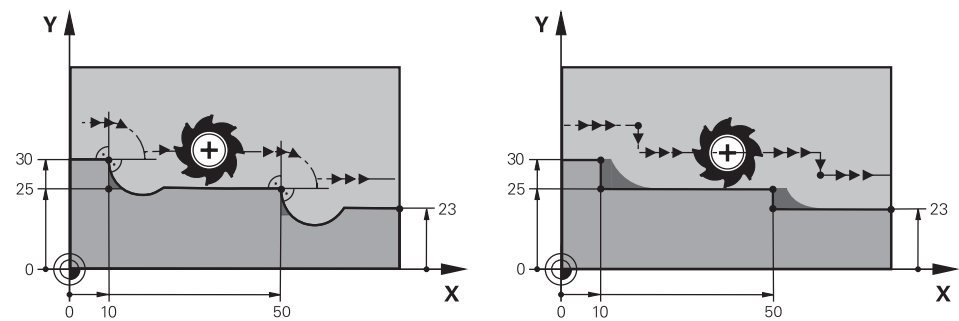
Contour step without **M97**

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.

18.4.3 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
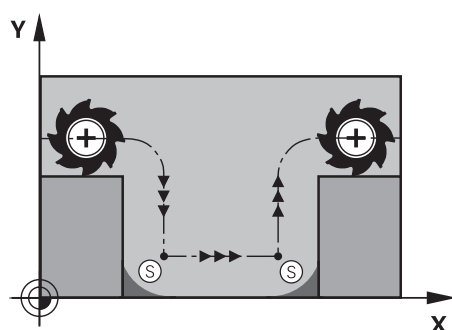
Open contour without **M98**
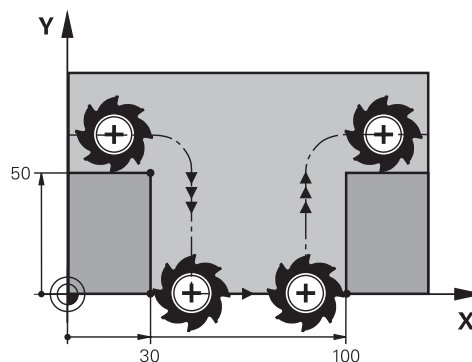
Open contour with **M98**

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.

18.4.4 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
13 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**.

18.4.5 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.

18.4.6 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.

18.4.7 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 Adv. Function 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 is also active if the **Tilt working plane** function is active. (#8 / #1-01-1)

  Further information: "Tilting the working plane (#8 / #1-01-1)", Page 343
- 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 582
- Without **M128** or **FUNCTION TCPM** (#9 / #4-01-1), **M116** can be in effect for multiple rotary axes at the same time.

18.4.8 Activating handwheel superimpositioning with M118

Application

With **M118** the control activates handwheel superimpositioning. You can then perform manual corrections by handwheel during program run.

Related topics

- Handwheel superimpositioning by means of the Global Program Settings (GPS (#44 / #1-06-1))

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

Requirements

- Handwheel

Description of function

Effect

M118 takes effect at the start of the block.

In order to reset **M118**, program **M118** without entering any axes.

> **i** Canceling a program also resets handwheel superimpositioning.

Application example

11 L Z+0 R0 F500	; Move in the tool axis
12 L X+200 R0 F250 M118 Z1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the Z axis

In the first NC block the control positions the tool in the tool axis.

In NC block **12** the control activates handwheel superimpositioning at the start of the block with a maximum traverse range of ±1 mm in the Z axis.

Then the control performs the traverse movement in the working plane. During this traverse movement you can use the handwheel for continuous motion of the tool in the Z axis by up to ±1 mm. This way you can, for example, rework a workpiece that has been reclamped but that cannot be probed due to its free-form surface.

Input

If you define **M118**, the control continues the dialog and prompts you for the axes and the maximum permissible superimpositioning value. For linear axes you define the value in millimeters and for rotary axes in degrees.

21 L X+0 Y+38.5 RL F125 M118 X1 Y1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the X and Y axes

Notes

> ⚙ Refer to your machine manual.
> Your machine manufacturer must have prepared the control for this function.

- By default **M118** is in effect in the machine coordinate system **M-CS**.

  When you activate the **Handwheel Superimpositioning** toggle switch in the **GPS** (#44 / #1-06-1) workspace, handwheel superimpositioning is active in the coordinate system that was selected most recently.

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

- 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.

Notes in conjunction with the virtual tool axis VT (#44 / #1-06-1)

> ⚙ Refer to your machine manual.
> Your machine manufacturer must have prepared the control for this function.

- On machines with head rotation axes, you can choose for inclined machining whether superimpositioning should be in effect in the Z axis or along the virtual tool axis **VT**.

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

- In the machine parameter **selectAxes** (no. 126203) the machine manufacturer defines the assignment of axis keys on the handwheel.

  When using an HR 5xx handwheel, you can assign the virtual axis to the orange **VI** axis key, if desired.

18.4.9 Pre-calculating a radius-compensated contour with M120

Application

With **M120** the control pre-calculates a radius-compensated contour. This way the control can produce contours that are smaller than the tool radius without damaging the contour or issuing an error message.

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
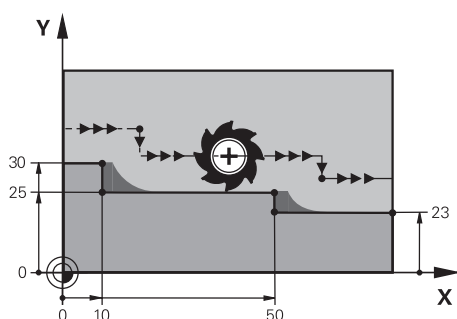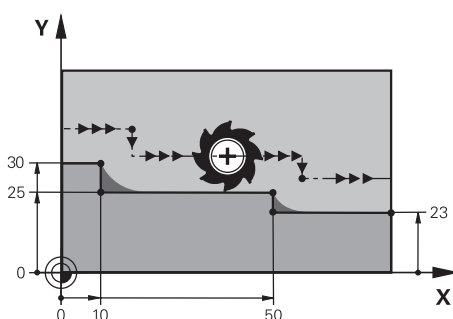
Contour step with **M97**   Contour step with **M120**

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
▶ Slowly prove-out the NC program

■ 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**

> **i** You can still execute NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Example
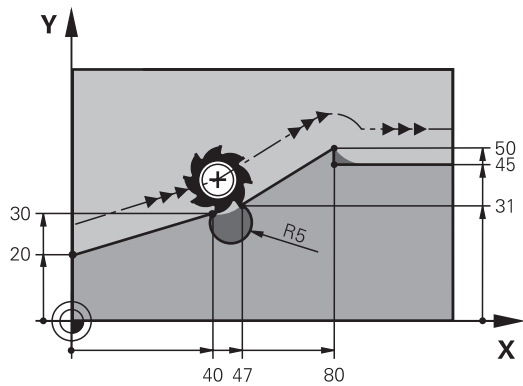
0 BEGIN PGM "M120" MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-10	
2 BLK FORM 0.2 X+110 Y+80 Z+0	; Workpiece blank definition
3 TOOL CALL 6 Z S1000 F1000	; Insert the tool with diameter 12
4 L X-5 Y+26 R0 FMAX M3	; Move in the working plane
5 L Z-5 R0 FMAX	; Infeed in the tool axis
6 L X+0 Y+20 RL F AUTO M120 LA5	; 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 run
14 END PGM "M120" MM	

Definition

Abbreviation	Definition
LA (look ahead)	Number of look-ahead blocks

18.4.10 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 384

- 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°.

18.4.11 Compensating the tool angle of inclination automatically with M128 (#9 / #4-01-1)

Application

If the position of a controlled rotary axis changes in the NC program, then the control uses **M128** during the tilting procedure to automatically compensate for the tool inclination with a compensating movement of the linear axes. That way the position of the tool tip relative to the workpiece surface remains unchanged (TCPM).

> **i** Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

Requirements

- Machine with rotary axes
- Kinematics description

  > **⚙** Refer to your machine manual.
  > The machine manufacturer creates the kinematics description of the machine.

- Software option Adv. Function Set 2 (#9 / #4-01-1)

Description of function

Effect

M128 takes effect at the start of the block.

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

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

> **i** **M128** is also in effect in the **Manual** operating mode and remains active even after a change in the operating mode.

Application example
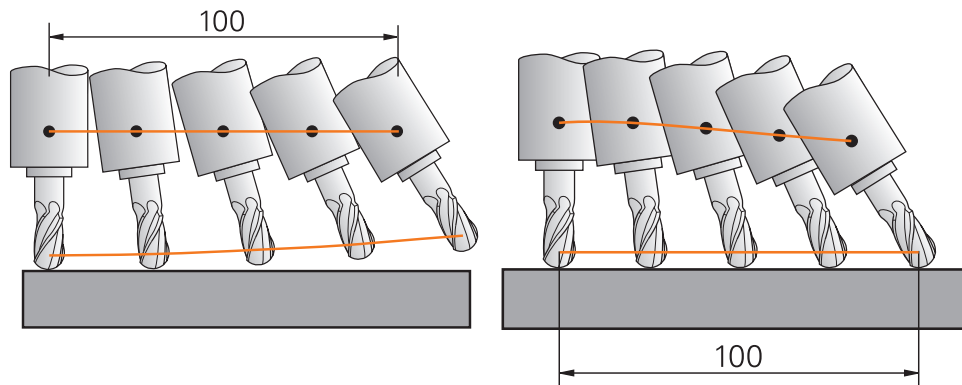
Behavior without **M128**

Behavior with **M128**

11 L X+100 B-30 F800 M128 F1000	; Move with automatic compensation of the motion in the rotary axis

In this NC block the control activates **M128** with the feed rate for the compensating movement. The control then simultaneously moves the tool in the X axis and in the B axis.

In order to keep the position of the tool tip constant relative to the workpiece while inclining the rotary axis, the control uses the linear axes to perform a continuous compensating movement. In this example the control performs the compensating movement in the **X** and **Z** axes.

Without **M128** an offset of the tool tip relative to the nominal position results as soon as the inclination angle of the tool changes. The control does not compensate for this offset. If you do not take this deviation into account in the NC program, the machining operation will not be performed correctly or a collision will occur.

Input

If you define **M128**, the control continues the dialog and prompts you for the feed rate **F**. The defined value limits the feed rate of the linear axis 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 199
> 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 positioning. There is a danger of collision while the axis moves out of the coupling and during the positioning 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. The control selects the tilting solution with the smallest number of rotary axis movements from the current position. This can result in unexpected traverse movements.
>
> ▶ Use the Simulation mode to test the NC program before execution
> ▶ Slowly prove-out the NC program

Further information: "3D tool compensation during peripheral milling (#9 / #4-01-1)", Page 435

Further information: "Output with vectors", Page 540

- 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

- If you always select the first selection option offered for **FUNCTION TCPM**, you will achieve the same functionality as with **M128**. In this case program the syntax **FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-TIP**.

- **M128** and **FUNCTION TCPM** with **AXIS POS** selected do not take an active basic rotation or 3D basic rotation into account. 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 393

- If **M128** is active, the control selects the tilting solution with the smallest number of rotary axis movements from the current position for **LN** straight lines.

- 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 310

- If you run the following functions while **M128** is active, then the control cancels program run and issues an error message:
  - Tool-tip radius compensation **RR/RL** in turning mode (#50 / #4-03-1)
  - **M91**

- **M92**
- **M144**
- Calling a tool with **TOOL CALL**
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1)) and simultaneous use of **M118**

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. For **FUNCTION TCPM** and **M128** the machine parameter applies only to one rotary axis of the table 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 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 316

  - 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 199

- 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 409

  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 468

- 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 393

Definition

Abbreviation	Definition
TCPM (tool center point management)	Maintain the position of the tool location point **Further information:** "Presets on the tool", Page 199

18.4.12 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, for example in conjunction with turning mode (#50 / #4-03-1).

Further information: "Switching the operating mode with FUNCTION MODE", Page 154

Description of function

Effect

M136 takes effect at the start of the block.

In order to reset **M136**, program **M137**.

Application example

11 LBL "TURN"	
12 FUNCTION MODE TURN	; Activate turning mode
13 M136	; Switch interpretation of the feed rate to mm/rev
14 LBL 0	

Here, **M136** is located in a subprogram in which the control activates turning mode (#50 / #4-03-1).

With **M136** the control interprets the feed rate as millimeters per spindle revolution, which is necessary for the turning mode. The feed rate per revolution refers to the rotational speed of the workpiece spindle. The control thus moves the tool at the programmed feed rate for every rotation of the workpiece spindle.

Without **M136** the control interprets the feed rate as millimeters per minute.

Notes

- In NC programs based on inch units, **M136** is not allowed in combination with **FU** or **FZ**.
- The workpiece spindle is not permitted to be controlled when **M136** is active.
- 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.

18.4.13 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 576

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

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (#9 / #4-01-1)", Page 393

- **PLANE** functions (#8 / #1-01-1)

  Further information: "Tilting the working plane with PLANE functions (#8 / #1-01-1)", Page 348

- 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 MOVE FMAX	; Tilt spatial angle **SPB** by 90°

On a six-axis machine with **A**, **B**, and **C** rotary axes you must exclude one rotary axis for spatial angle operations; otherwise too many combinations are possible.

With **M138 A C** the control calculates the axis position when tilting with spatial angles only in the **A** and **C** axes. The B axis is excluded. Therefore, in NC block **12** the control positions the spatial angle **SPB+90** with the **A** and **C** axes.

Without **M138** there are too many possibilities for tilting. The control interrupts the machining process and issues an error message.

Input

If you define **M138**, the control continues the dialog and prompts you for the rotary axes to be taken into account.

11 L Z+100 R0 FMAX M138 C	; Define that the C axis should be taken into account

Notes

- With **M138** the control excludes the rotary axes only during the calculation and positioning of spatial angles. A rotary axis that has been excluded with **M138** can nevertheless be moved in a positioning block. Please note that in this case the control does not execute any compensations.
- In the optional machine parameter **parAxComp** (no. 300205) the machine manufacturer defines whether the control includes the position of the excluded axis when calculating the kinematics.

18.4.14 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 323
- 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

18.4.15 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

18.4.16 Taking the tool offset into account in calculations with 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.

You can use **M144** (#50 / #4-03-1) for an inclined turning operation, for example.

> **i** HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM** (#9 / #4-01-1) instead of **M144**.
>
> Exceptions are, e.g.:
> - **RL** or **RR** positioning blocks with tool radius compensation
> - Positioning blocks with **M91**
> - Positioning blocks with tool tip radius compensation **SRK** (#50 / #4-03-1)

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 393
- Inclined turning (#50 / #4-03-1)

  Further information: "Inclined turning", Page 162

Requirement

- Software option Adv. Function 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.

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 558

- 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 348

  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 310

Notes on turning (#50 / #4-03-1)

- If the inclined axis is a tilting table, the control changes the orientation of the workpiece coordinate system **W-CS** versus the machine coordinate system **M-CS**.

  If the tilted axis is a swivel head, the control changes the orientation of the tool coordinate system **T-CS** versus the machine coordinate system **M-CS**.

  Further information: "Notes concerning different machine kinematics", Page 344

- After inclining a rotary axis, you possibly have to pre-position the turning tool in the Y coordinate and orient the position of the tool tip with Cycle **800 ADJUST XZ SYSTEM**.

  Further information: User's Manual for Machining Cycles

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

Application

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

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

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

Related topics

- Automatic retraction with **FUNCTION LIFTOFF**
  Further information: "Automatic tool liftoff with FUNCTION LIFTOFF", Page 482

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.

18.4.18 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
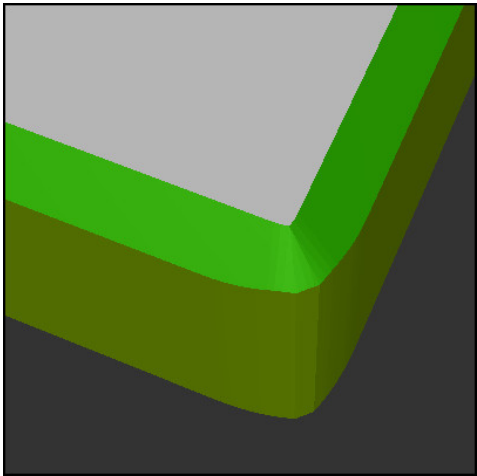
Contour without **M197**
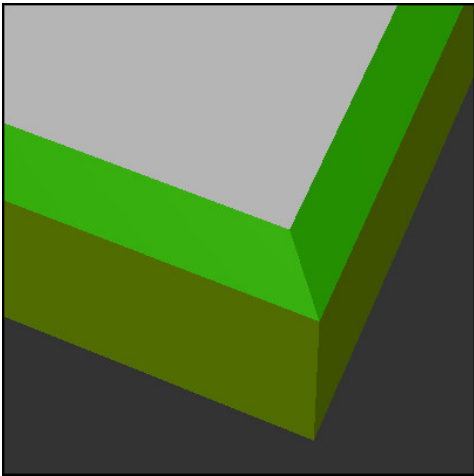
Contour with **M197**

* - ...	; 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

18.5 Miscellaneous functions for tools

18.5.1 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

> **i** Use only tools with an identical radius as replacement tools. The control does not automatically check the radius of the tool.
>
> If you want the control to check the radius, program **M108** after the tool change.
>
> **Further information:** "Checking the radius of the replacement tool with M108", Page 595

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!

- The **M101** miscellaneous function is not available for turning tools and in turning mode (#50 / #4-03-1).

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.

18.5.2 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 424

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 539

Requirement

- Software option Adv. Function 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

DR2+
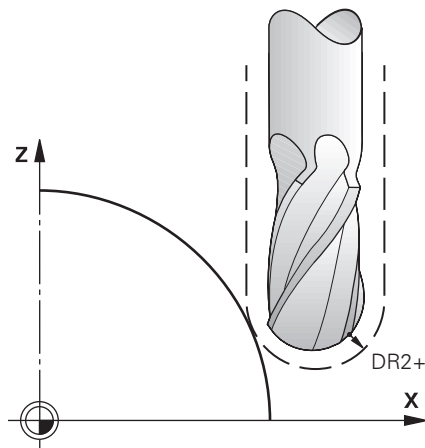
11 TOOL CALL 1 Z S5000 DR2:+0.3	; Insert a tool with a positive delta value
12 M107	; Permit positive delta values

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 435
- 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 428

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

18.5.3 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 591

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 593

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

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 **Single Block** mode

19

Programming with variables

19.1 Overview of variable programming

Variables are placeholders for numbers and texts which can take on different values.

You can use the variables to, for example, carry out calculations or create variable logs in the NC program.

The control provides the following options for variable programming:

Topic	Further information
Variables	Page 599
String functions	Page 642
Format strings	Page 649
Defining the counter with **FUNCTION COUNT**	Page 653
Program defaults for cycles	See the User's Manual for Machining Cycles
Table access with SQL statements	Page 655

19.2 Variables: Q, QL, QR, QS parameters and named parameters

19.2.1 Basics

Application

You can use the Q, QL, QR, QS and named 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.
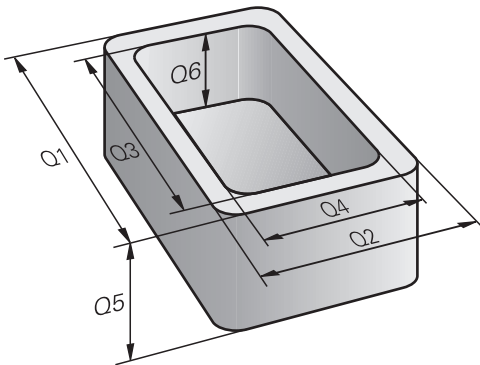
Description of function

Variables are composed of the variable name and the variable value.

The control provides the following types of variables:

Variable type	Category	Example	Further information
Q parameters	Numerical parameter	**Q10 = +10**	Page 601
QL parameters	Numerical parameter	**QL10 = +10**	Page 601
QR parameters	Numerical parameter	**QR10 = +10**	Page 601
QS parameters	String parameters	**QS10 = "123"**	Page 601
Named parameters	Numerical parameter or String parameters	**{DEPTH} = -10** or **{TOOL} = "MILL_D8"**	Page 602

Variable name

The control displays the variable name to the left of the equal sign.

For the different variable types, the variable name is as follows:

Variable	Contents
Q, QL, QR or QS parameters	The variable name of these parameters consists of letters and numbers (e.g., **Q10** or **QS10**).
	The control defines the letters for the variable type.
Named parameter	The variable name of named parameters consists of a freely chosen designation enclosed in curly brackets (e.g., **{DEPTH_1}**).
	The variable name can include letters, numbers and underscores, but it must start with a letter.
	For named parameters you can define a variable name with up to 31 characters.

Variable value

The control shows the variable value to the right of the equal sign.

The possible variable value differs as follows, depending on the category:

Category	Contents
Numerical parameter	Numerical parameters can be assigned a variable value between -999 999 999 and +999 999 999.
	The input range is limited to no more than 16 characters; up to nine of these characters can precede the comma. The control can calculate numbers up to a magnitude of 10^{10}.
String parameters	String parameters can be assigned a variable value of up to 255 characters.
	The control displays the variable values of string parameters in quotation marks (e.g., **"TOOL_3"**).
	The following characters are allowed in the variable value of string 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 ; ! # $ % & ' ( ) + , - . / : < = > ? @ [ ] ^ _ ` *

> **i** The control provides the **FMT** syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings.
>
> **Further information:** "Format strings", Page 649

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 in 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 1599	Q parameters for functions defined by the machine manufacturer (e.g., cycles)
1600 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 parameters 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.

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

Named parameter

Named parameters, just like QL parameters, take effect locally in the NC program.

You can define named parameters as numerical parameters or as string parameters.

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 606

- You can enter fixed and variable values mixed in the NC program.
- 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 729

- With the **SET UNDEFINED** syntax element you assign the **undefined** status to the variables.

  For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

  If you use an undefined variable in arithmetic operations in the NC program, the control displays an error message and stops program run.

  Further information: "Assigning the Undefined status to a variable", Page 615

- 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.

■ For each variable type, you can define the variable range or variables that the control will display on the **QPARA** tab of the **Status** workspace.

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

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

19.2.2 The Q parameter list window

Application

In the **Q parameter list** window, you can view and edit all values of all variables, if required.

Related topics

■ Fundamentals of variables
Further information: "Basics", Page 599
■ The **QPARA** tab in the **Status** workspace
Further information: User's Manual for Setup and Program Run

Description of function

You can open the **Q parameter list** window with the **Q info** button in the following operating modes:

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

In the **Manual** operating mode, you can also use the **Q** key to open the window.
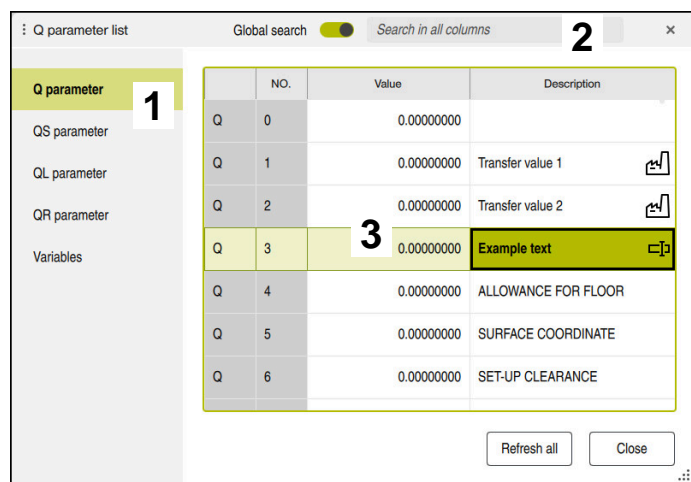
The **Q parameter list** window with the Q parameter values

The **Q parameter list** window shows the following areas:

1 Variable type

You can select the variable type shown by the control such as Q parameters.

2 Search

By default, the control searches in all columns. If you deactivate the **Global search** switch, the search will be restricted to the currently selected column.

You can begin the search by entering a character as soon as the **Q parameter list** window opens.

3 Contents

Depending on the variable type, the control shows the following information:

- Variable type
- Number or name
- Value

  If the cell in the **Value** column is highlighted in white, you can edit its value.

- Description

  Texts that are preassigned by HEIDENHAIN, defined by the machine manufacturer or entered by the operator

  Further information: "Options for descriptive texts", Page 605

Options for descriptive texts

Upon a long-press gesture or a right-click the control offers the following options for descriptive texts:

- **Default configuration**

  The control shows the descriptive text preassigned by HEIDENHAIN.

- **Machine manufacturer configuration**

  The control shows the language-sensitive descriptive text defined by the machine manufacturer.

  This selection is available only if the machine manufacturer has stored a descriptive text for this variable.

- **Edit**

  The control opens the input field and you can enter a descriptive text.

  You can also activate the input field with a double-tap or double-click.

If the machine manufacturer has stored descriptive texts, the control shows the **Machine manufacturer configuration** by default.

The control displays the selected text until you select a different option.

Icons

The **Q parameter list** window contains the following icons:

Icon	Meaning
🏭	Descriptive text of machine manufacturer is selected
⊏⌷	User-defined descriptive text is selected

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- 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.

  The control reaches this required status after, for example, an NC block has been executed in **Single Block** mode.

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

- 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.

- If you select or enter a different descriptive text, the control does not store the present user-defined descriptive texts.

19.2.3 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 124

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 553

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 726

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

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 parameter's **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 Q168

The control assigns the calculated deviation values to the Q parameters **Q161** to **Q168**:

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
Q168	Deviation from **Q160 COORD., MEASURING AXIS**

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

19.2.4 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 606

- Calculations using formulas

  Further information: "Formulas in the NC program", Page 636

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
$\sqrt{}$	**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 numbers 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
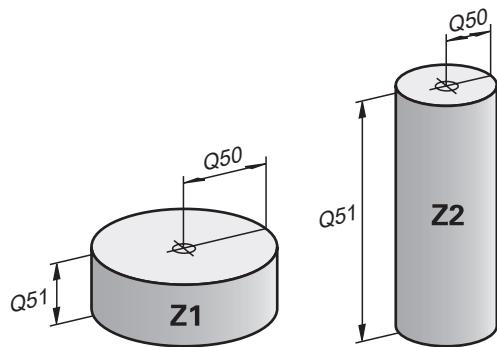
Cylinder radius:	R = Q50
Cylinder height:	H = Q51
Cylinder Z1:	Q50 = +30
	Q51 = +10
Cylinder Z2:	Q50 = +10
	Q51 = +50

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**).

19.2.5 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 numbers in the equations can be entered with an algebraic sign.

Definition
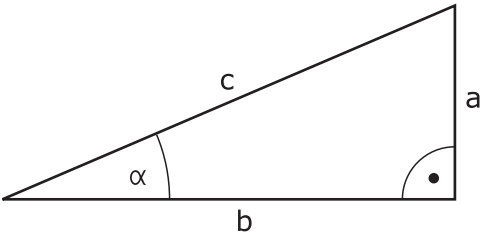
Side or trigono-metric function	Meaning
a	Opposite side The side opposite to angle **α**
b	Adjacent side The side adjacent to angle **α**
c	Hypotenuse The longest side of the triangle, opposite to the right angle
Sine	sin **α** = opposite side/hypotenuse sin **α** = a/c
Cosine	cos **α** = adjacent side/hypotenuse cos **α** = b/c
Tangent	tan **α** = opposite side/adjacent side tan **α** = a/b or tan **α** = sin **α**/cos **α**
Arc tangent	**α** = arctan(a/b) or **α** = arctan(sin **α**/cos **α**)

Example

a = 25 mm

b = 50 mm

α = arctan(a/b) = arctan 0.5 = 26.57°

Furthermore:

a²+b² = c² (where a² = 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

19.2.6 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 124

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**

> **ℹ** 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.

19.2.7   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 282

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 284

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

19.2.8 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 messages output with **FUNCTION REPORT**

  Further information: "Issuing messages with FUNCTION REPORT", Page 759
- Error messages in the notification menu

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

---

📖 **Overview of the Machine Parameters, Error Numbers and System Data**

The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:

- Machine parameters of the **MPs for setters** application
- Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
- System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

ID 1445456-xx

You can download this documentation free of charge from the HEIDENHAIN website.

TNCguide

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 error numbers from 0 to 999 as well as those from 3000 to 9999 are defined by the machine manufacturer.

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	Syntax initiator for error message output
Number	Number of the error message
	Number or numerical parameter

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 **FN 16: F-PRINT** function you can output formatted fixed and variable 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:

- Format file

  The format 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 622

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 624

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 459

Formatting

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

> ℹ️ Please note that the input is case-sensitive.

Formatting characters	Meaning
"…"	Identify the formatting of the contents to be output
	ℹ️ For text output, you can use the UTF-8 character set.
%F, **%D** or **%I**	Initiate the formatted output of numerical 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 string 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_JAPANESE	Output text only for Japanese 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	Syntax initiator for formatted output of contents
File	Path of the format 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

> **i** If you want to replace the content of the window for multiple screen outputs in the NC program, define the **M_CLOSE** or **M_TRUNCATE** keyword.

The control opens the **FN16-PRINT** window for screen output. The window remains open until you close it. While the window is open, you can operate the control in the background and change to another operating mode.

You can close the window in the following ways:

- Define 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 format 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.

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 format 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 139

  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 a text and assign it to the defined character 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**:

⋮ FN 16 output

TOUCHPROBE

POS1: 100

DATE: 08.08.2024
TIME: 14:09

OK
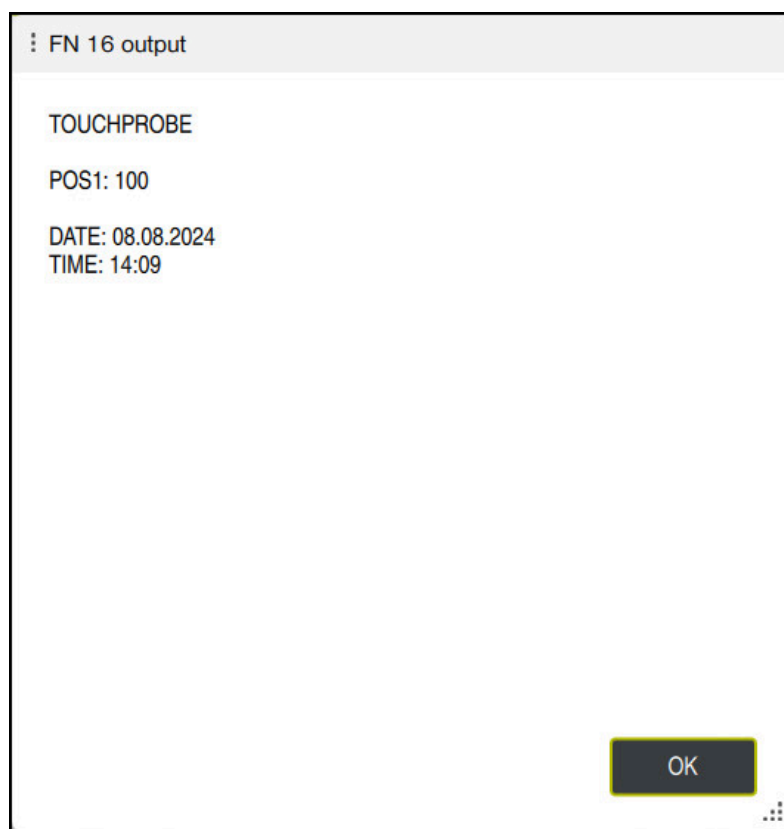
The **FN16-PRINT** window

Read system data with FN 18: SYSREAD

Application

With the **FN 18: SYSREAD** function you can read numerical system data and store this data in variables.

Related topics

- Reading system data using string parameters

  Further information: "Read system data with SYSSTR", Page 643

> 📖 **Overview of the Machine Parameters, Error Numbers and System Data**
>
> The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:
>
> - Machine parameters of the **MPs for setters** application
> - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
> - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions
>
> ID 1445456-xx
>
> You can download this documentation free of charge from the HEIDENHAIN website.
>
> **TNCguide**

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	Syntax initiator for reading system data
Parameter	Numerical parameter in which the control stores the information
ID	Group number of the system datum Number or numerical parameter
NR	System data number Number or numerical parameter Optional syntax element
IDX	Index Number or numerical parameter Optional syntax element
.	Sub-index for system data for tools Number or numerical parameter 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 824

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	Syntax initiator for sending information
Name or **Parameter**	Format of the text to be transmitted
	Text or string parameter
	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 622
/	Contents of the up to seven placeholders in the output text
	Number or numerical parameter
	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"	; Amount passed (OK) absolute
12 FN 38: SEND /"JOB:1234_STEP:1_OK_I:1"	; Amount passed (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

19.2.9 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 827

- Access to table values in case of low computing power

  Further information: "Table access with SQL statements", Page 655

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	Syntax initiator for opening a table
File	Path of the table to be opened
	Text or string parameter
	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 827

- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 633

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. The contents to be written to the columns must have been defined previously, using variables, or are defined directly in the **FN 27** NC function.

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	Syntax initiator for writing to a table
Number	Row number of the table to be written to Number or numerical parameter
Name or **Parameter**	Column names in the table to be written to Text or string parameter 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 **Parameter**	Table value Number, text, or variable 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.
- When you write into several columns, the control can only write either numbers or texts.
- If you define a fixed value in the **FN 27** NC function, the control will write the same value to each defined column.
- With the **SET UNDEFINED** syntax element you assign the **undefined** status to the variables.

  For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

  If you use an undefined variable in arithmetic operations in the NC program, the control displays an error message and stops program run.

  Further information: "Assigning the Undefined status to a variable", Page 615

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 827

- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 633

- Writing a freely definable table

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 633

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	Syntax initiator for reading from a table
Parameter	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
	Number or numerical parameter
Name or **Parameter**	Column name in the table to be read
	Text or string parameter
	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 text from column **DOC**

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.

19.2.10 Formulas in the NC program

Application

The NC functions **Formula** or **Variable** allow you to define several arithmetic operations in one NC block. You can also assign a single value to a variable.

Related topics

■ String formula for texts

Further information: "String functions", Page 642

■ Defining a single calculation in an NC block

Further information: "The Basic arithmetic folder", Page 613

Description of function

The **Formula** NC function is used to define the **Q**, **QL** and **QR** parameters.

Use the **Variable** NC function to define a named parameter.

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 640

■ 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 729

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: "Examples", Page 640

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 638

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 **SQ**	**Square** (square) Example: **Q15 = SQ 5**	Function
SQRT **SQRT**	**Calculate square root** (square root) Example: **Q22 = SQRT 25**	Function
SIN **SIN**	**Calculate sine** Example: **Q44 = SIN 45**	Function
COS **COS**	**Calculate cosine** Example: **Q45 = COS 45**	Function
TAN **TAN**	**Calculate tangent** Example: **Q46 = TAN 45**	Function
ASIN **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 **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 **ATAN**	**Calculate arctangent** Inverse function of tangent The control determines the angle from the ratio of the opposite side to the adjacent side. Example: **Q12 = ATAN Q50**	Function

Button	Arithmetic operation	Operator
^	**Exponentiation** Example: **Q15 = 3 ^ 3**	Power
PI	**Use the "pi" constant** π = 3.14159 Example: **Q15 = PI**	
LN	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG	**Negate** Multiply by −1 Example: **Q2 = NEG Q1**	Function
INT	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42** ⓘ The **INT** function does not round off—it simply truncates the decimal places. Input: **0...999999999**	Function
ABS	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN	**Check the algebraic sign** Example: **Q12 = SGN Q50** If **Q50 = 0**, then **SGN Q50 = 0** If **Q50 < 0**, then **SGN Q50 = -1** If **Q50 > 0**, then **SGN Q50 = 1**	Function
%	**Calculate the modulo value (division remainder)** Example: **Q12 = 400 % 360** Result: **Q12 = 40**	Function

Further information: "The Basic arithmetic folder", Page 613
Further information: "The Trigonometric functions folder", Page 616

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** or **Variable**
▶ Define a variable name 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

Examples

Multiplication and division before addition and subtraction

| **11 Q1 = 5 * 3 + 2 * 10** | ; Result = 35 |

| **11 {a} = 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 |

| **11 {b} = 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 |

| **11 {c} = 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

11 {d} = SIN (50 - 20)	; Result = 0.5

- 1st calculation: Perform operations in parentheses first: 50 − 20 = 30
- 2nd calculation: Calculate sine of 30 = 0.5

19.3 String functions

Application

The string functions allows you to define and process texts using string parameters, for example, in order to create variable logs with **FN 16: F-PRINT**. In computing, a string designates a text.

Related topics

■ Ranges of variables

Further information: "Basics", Page 599

■ Programming of format strings

Further information: "Format strings", Page 649

Description of function

Within the **Formula**, **Variable** or **String formula** NC functions you can use the following string functions:

Syntax element	Meaning	NC function
DECLARE STRING	Assign a text to a QS parameter **Further information:** "Assigning text to a string parameter", Page 644	**DECLARE STRING**
| |	Concatenate contents of string parameters and assign them to a string parameter **Further information:** "Concatenating values of string parameters", Page 645	■ **String formula** ■ **Variable**
TONUMB	Convert the value of a QS parameter to a numerical value and assign it to a numerical parameter **Further information:** "Converting the value of a string parameter to number ", Page 645	■ **Formula** ■ **Variable**
TOCHAR	Convert numerical value to a text and assign it to a string parameter **Further information:** "Converting numerical values to texts", Page 646	■ **String formula** ■ **Variable**
SUBSTR	Copy a substring from a QS parameter and assign it to a string parameter **Further information:** "Copying a substring from a string parameter", Page 646	■ **String formula** ■ **Variable**
SYSSTR	Read system data and assign the contents to a string parameter **Further information:** "Read system data with SYSSTR", Page 643	■ **String formula** ■ **Variable**
INSTR	Search for a substring in a QS parameter and assign the retrieved characters to a numerical parameter **Further information:** "Searching for a substring within QS parameter contents", Page 646	■ **Formula** ■ **Variable**
STRLEN	Determine the string length of a QS parameter and assign it to a numerical parameter **Further information:** "Determining the number of characters in QS parameter contents", Page 647	■ **Formula** ■ **Variable**

Syntax element	Meaning	NC function
STRCOMP	Compare QS parameters in ascending lexical order and assign the result to a numerical parameter **Further information:** "Comparing the lexical order of two texts", Page 647	■ **Formula** ■ **Variable**
CFGREAD	Read the content of a machine parameter and assign it to a variable **Further information:** "Applying the contents of a machine parameter", Page 648	■ **String formula** ■ **Formula** ■ **Variable**

The control provides the following options to enter formulas:

■ Auto-complete

Further information: "Entering a formula using the auto-complete function", Page 640

■ 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 729

Read system data with SYSSTR

With the **SYSSTR** NC function you can read system data and save the contents in string parameters. Select the system datum by means of a group number **ID** and a number **NR**. Optionally, you can enter **IDX** and **DAT**.

SYSSTR only allows you to read alphanumeric values.

For numerical values, the control offers the **FN 18: SYSREAD** NC function. **SYSSTR** is programmed identically to **FN 18: SYSREAD**.

Further information: "Read system data with FN 18: SYSREAD", Page 629

> **Overview of the Machine Parameters, Error Numbers and System Data**
>
> The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:
>
> ■ Machine parameters of the **MPs for setters** application
>
> ■ Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
>
> ■ System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions
>
> ID 1445456-xx
>
> You can download this documentation free of charge from the HEIDENHAIN website.
>
> **TNCguide**

19.3.1 Assigning text to a string parameter

Before you can use and process texts, you must assign characters to the string parameters.

To assign a text to a string parameter:

Insert
NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **String formula** or **Variable**
- ▶ Define a variable name for the result
- ▶ Move cursor to the right
- ▶ If required, enter **""**
- ▶ Enter the desired value between the quotation marks
- ▶ End the NC block
- ▶ Execute the NC block
- > The control saves the entered value in the target parameter.

> **i** Alternatively, you can assign characters to a QS parameter using the **DECLARE STRING** NC function.

In these examples, the control assigns a text to a string parameter.

| **11 QS10 = "workpiece"** | ; Assign text to the QS parameter **QS10** |

| **11 {a} = "workpiece"** | ; Assign text to the named parameter **{a}** |

19.3.2 Concatenating values of string parameters

With the **||** concatenation operator, you can concatenate the contents of multiple string parameters. This allows you, for example, to combine fixed and variable texts.

To concatenate the contents of multiple string parameters:

> Insert
> NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **String formula** or **Variable**
- ▶ Define a string parameter for the result
- ▶ Confirm your input

> ⌫

- ▶ Select the backspace key, if required
- > The control deletes the quotation marks.
- ▶ Select **QS** or **Variable**
- ▶ Enter the variable name
- ▶ Press the spacebar
- > The control displays the currently available syntax elements.
- ▶ Select concatenation operator **||**
- ▶ Select **QS** or **Variable**
- ▶ Enter the variable name
- ▶ End NC block
- > After their execution, the control saves the substrings in succession as text in the target parameter.

In these examples, the control concatenates the contents of two string parameters and assigns the result to a third string parameter.

Parameter contents:

- **QS12** and **{b}**: **Status:**
- **QS13** and **{c}**: **Scrap**
- **QS10** and **{a}**: **Status: Scrap**

11 QS10 = QS12 || QS13	; Concatenate contents of **QS12** and **QS13** and assign them to the QS parameter **QS10**

11 {a} = {b} || {c}	; Concatenate contents from **{b}** and **{c}** and assign them to named parameter **{a}**

19.3.3 Converting the value of a string parameter to number

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 these examples, the control converts the variable value of a QS parameter to a numerical value. The control assigns this value to a numerical parameter.

11 Q82 = TONUMB (SRC_QS11)	; Convert value from **QS11** to a numerical value and assign it to **Q82**

11 {a} = TONUMB (SRC_QS11)	; Convert value from **QS11** to a numerical value and assign it to **{a}**

19.3.4 Converting numerical values to texts

With the **TOCHAR** NC function, you can save the content of a numerical parameter in a string parameter. You can concatenate the saved content with other string parameters, for example.

In these examples, the control converts the numerical value of a Q parameter to a text. The control assigns this text to a string parameter.

11 QS11 = TOCHAR (DAT+Q50 DECIMALS3)	; Convert numerical value from **Q50** to a text and assign it to the QS parameter **QS11**

11 {a} = TOCHAR (DAT+Q50 DECIMALS3)	; Convert numerical value from **Q50** to a text and assign it to the named parameter **{a}**

19.3.5 Copying a substring from a string parameter

With the **SUBSTR** NC function, you can save a defined substring from a QS parameter in another string parameter. For example, you can use this NC function to extract the file name from an absolute file path.

In these examples, the control saves the substring of a QS parameter in another string parameter. 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**

11 {a} = SUBSTR (SRC_QS10 BEG2 LEN4)	; Assign substring from **QS10** to the named parameter **{a}**

19.3.6 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 a QS parameter for the text of a second QS parameter. When counting the characters, the control starts from zero. The control assigns the occurrence to the numerical parameter as a number.

37 Q50 = INSTR (SRC_QS10 SEA_QS13 BEG2)	; Search **QS10** for substring from **QS13**

19.3.7 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 a QS parameter. The control assigns the determined number to a numerical parameter.

11 Q52 = STRLEN (SRC_QS15)	; Determine the number of characters in **QS15** and assign it to **Q52**

19.3.8 Comparing the lexical order of two texts

With the **STRCOMP** NC function, you can compare the lexical order of the content of two QS parameters.

The control returns the following results:

- **0**: The content of the two parameters is identical
- **-1**: In the lexical order, the content of the first QS parameter comes **before** the content of the second QS parameter
- **+1**: In the lexical order, the content of the first QS parameter comes **after** the content of the second QS parameter

The lexical order is as follows:
1 Special characters (e.g., ?_)
2 Numerals (e.g., 123)
3 Uppercase letters (e.g., ABC)
4 Lowercase letters (e.g., abc)

> **i** Starting from the first character, the control proceeds until the contents of the QS parameters differ from each other. If the contents differ starting from, for example, the fourth digit, the control aborts the check at this point.
>
> Shorter contents with identical strings are displayed first in the order (e.g., abc before abcd).

In this example, the control compares the lexical order of the values of two QS parameters. The control assigns the result to a numerical parameter as a number.

11 Q52 = STRCOMP (SRC_QS12 SEA_QS14)	; Compare the lexical order of the values of **QS12** and **QS14**

19.3.9 Applying the contents of a machine parameter

The NC function **CFGREAD** allows you to read out the values from machine parameters.

Before using **CFGREAD**, you must program one QS parameter each with key, entity and attribute. Pay attention to enter the correct upper and lower case letters.

The NC function includes the following syntax elements:

Syntax element	Meaning
KEY_QS	QS parameter with the group name of the machine parameter (key)
	If no key is available, define the QS parameter without content.
TAG_QS	QS parameter with the object name of the machine parameter (entity)
ATR_QS	QS parameter with the name of the machine parameter (attribute)
IDX	Index of the machine parameter
	Number or numerical parameter
	Optional syntax element

> **i** You determine the required values using the table view of the configuration editor.
> **Further information:** User's Manual for Setup and Program Run

If the machine parameter contains numerical values, you can use the values in Q, QL, or QR parameters. The control always outputs numerical values in the metric system. Strings can be transferred in QS parameters. Named parameters allow the transfer of numerical values and strings.

Examples

In this example you transfer the overlap factor from the machine parameter **pocketOverlap** (no. 201001):

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

In this example you transfer the position of the tool touch probe in the Y axis (index 1) from the machine parameter **centerPos** (no. 114313):

11 QS11 = "TT140_2"	; Assign the key to the QS parameter **QS11**
12 QS12 = "CfgTTRectStylus"	; Assign the entity to the QS parameter **QS12**
13 QS13 = "centerPos"	; Assign the attribute to the QS parameter **QS13**
14 {a} = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX1)	; Read out the contents of the machine parameter

Note

If you use the **String formula** NC function, the result will always be a text. If you use the **Formula** NC function, the result will always be a numerical value.

19.4 Format strings

Application

The control provides the **FMT** syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings.

You can use format strings in the following NC functions, for example:

- **String formula**
- **SQL SELECT**
- **TEXT** within **FUNCTION REPORT**

Related topics

- Types of variables
  Further information: "Basics", Page 599
- Assign text to a string parameter
  Further information: "Assigning text to a string parameter", Page 644
- Concatenate values of string parameters
  Further information: "Concatenating values of string parameters", Page 645

Requirements

- Code number 555343
- Variables defined beforehand that are programmed within a format string

Description of function

The **FMT** syntax element allows you to define format strings. Program format strings in the text mode of the **Program** workspace.

Further information: "Text mode", Page 150

The syntax of the format strings is based on the f strings in Python. After the syntax initiator follows the string between double quotation marks. You can embed variables in the string using curly brackets. Once the NC block has been executed, the string contains the values of the embedded variables.

11 Q1 = +7	; Numerical value
12 QS1 = "TNC"	; Alphanumeric value
13 QS2 = FMT"ENJOY {QS1}{Q1}"	; Result after execution: **ENJOY TNC7**

Influencing the formatting of the string is also possible (e.g., by defining whether or not and how the control outputs algebraic signs). You define the formatting inside the curly brackets after a colon.

| **11 Q1 = +7** | ; Numerical value |
| **12 QS1 = FMT"{Q1:+}"** | ; Result including algebraic sign after execution:
+7 |

Formatting

A variety of formatting options is available, depending on the variable type and the values.

When you program formatting options, you must observe the following sequence:
[[fill character] alignment] [sign] ['0'] [field width] ['.' accuracy] [type]

> **ⓘ**   ■ [ ] are not programmed and only serve as separators within the sequence.
>   ■ ' ' are not programmed, however they contain the character to be programmed or an example input.
>   ■ '0' between the sign and the field width can be used as a fill character for numerical values.

11 QS2 = FMT"{QS1:X>10}"	; [[Fill character] alignment] [field width]
12 QS3 = FMT"{Q1:+.2f}"	; [Sign] ['.' accuracy] [type]

Fill character, alignment and field width

Formatting character or example	Meaning
[Fill character] (e.g., '**X**')	In connection with alignment and field width you can optionally define fill characters that replace any existing blank spaces.
<	Left-aligns the text within the available field width Default setting that is also effective without input
>	Right-aligns the text within the available field width
^	Centers the text within the available field width
[Field width] (e.g., '**10**')	You can define the field width as the number of places. If you define a field width that is smaller than the number of characters of the string, the value will have no function. If you do not define a value, the content will determine the field width.

11 QS1 = "LEFT"	; Alphanumeric value
12 QS2 = "RIGHT"	; Alphanumeric value
13 QS3 = FMT"{QS1:<4}{QS2:>6}"	; Result including algebraic sign after execution: **LEFT RIGHT**
14 QS4 = FMT"{QS1:X>10}"	; Result including algebraic sign after execution: **XXXXXXLEFT**

Algebraic sign

Formatting characters	Meaning
+	Outputs the sign for negative values as well as positive values
-	Outputs the sign for negative values only Default setting that is also effective without input
Spaces	If a space character is programmed as a formatting character, positive values will induce the control to output a leading space character.

11 Q1 = +7	; Numerical value
12 QS1 = FMT"{Q1}{Q1:+}"	; Result after execution: **7+7**

Integers

Formatting characters	Meaning
d	Outputs a decimal integer
	If you program this formatting for a floating-point number, the control will abort program run with an error message.
	If you do not define a type, the control will use the default setting for decimal numbers.

11 Q1 = +1	; Numerical value is integer
12 QS1 = FMT"{Q1:d}"	; Result after execution: **1**
13 Q1 = +1.23	; Numerical value is decimal number
14 QS1 = FMT"{Q1:d}"	; Program cancellation with error message

Decimal numbers

Formatting characters	Meaning
[Accuracy] (e.g., '**.2**')	You can define the accuracy of the output (e.g., by the number of decimal places for fixed-point numbers **f**).
	The effect of the accuracy varies, depending on the combination with the following formatting signs.
e	Outputs the exponent representation with **e** as a separator
E	Outputs the exponent representation with **E** as a separator
f	Outputs a decimal fixed-point number
	Define the number of decimal places using the accuracy.
	If you do not define an accuracy, the control will use the default value '**6**'.
g	Depending on the value, the control automatically decides on the type of representation, for example, the exponent representation using **e** as separator.
	An additional accuracy allows you to define the number of significant digits to which the value is rounded by the control.
	If you do not define an accuracy, the control will use the default value '**6**'.
G	Like type **g**, but using **E** as separator for the exponent representation
Default [type]	If you program neither type nor accuracy, the control will use type **g** with an unlimited accuracy.
	The default type '**6**' does not apply in this case.

11 Q1 = +1.23	; Numerical value
12 QS1 = FMT"{Q1:e}"	; Result after execution: **1.230000e+00**
13 QS1 = FMT"{Q1:E}"	; Result after execution: **1.230000E+00**
14 QS1 = FMT"{Q1:f}"	; Result after execution: **1.23**

15 Q1 = +0.0000123	; Numerical value
16 QS1 = FMT"{Q1:f}"	; Result after execution: **0.0000123**
17 QS1 = FMT"{Q1:g}"	; Result after execution: **1.23e-05**
18 QS1 = FMT"{Q1:G}"	; Result after execution: **1.23E-05**
19 Q1 = +123.456	; Numerical value
20 QS1 = FMT"{Q1:.2}"	; Result after execution: **1.2e+02**
21 QS1 = FMT"{Q1:.2f}"	; Result after execution: **123.46**
22 QS1 = FMT"{Q1:.2g}"	; Result after execution: **1.2e+02**
23 QS1 = FMT"{Q1:.3}"	; Result after execution: **123**

Definition

Significant digits

Significant digits are the places of a number which contain valuable information. These digits start from the first number after the leading zeros, if applicable, and end with the last still meaningful number. For example, you can restrict a probing result to a number of significant digits by programming an accuracy.

Notes

- The control exclusively supports the described formatting options.
- If you, for example, apply a formatting for decimal numbers to texts, the control will abort the program run with an error message.

19.5 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 of workpieces up to which the control repeats the NC program.

Description of function

The control takes the **FUNCTION COUNT** function into consideration during program run and in the simulation.

The control uses a separate counter each for program run and for the simulation.

The control shows the current counter reading and the defined target count of machining operations on the **PGM** tab of the **Status** workspace.

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

The counter reading remains the same after a restart of the control.

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 653

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 Number, text, or variable Input: **0...9999**
SET	Assign a defined value to the counter Number, text, or variable Input: **0...9999**
TARGET	Define the target count to be reached Number, text, or variable Input: **0...9999**
REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet Number, text, or variable

Notes

> ***NOTICE***
>
> **Caution: Data may be lost!**
>
> The control manages the same counter for the **Program Run** operating mode and for the **MDI** application. The counter applies to all programs. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted, if applicable.
>
> ▶ Please check prior to machining whether a counter is active.

- On the **PGM** tab of the **Status** workspace you also find the **Counter settings** window, which allows you to define the counter.

  When the control executes **FUNCTION COUNT** in the NC program, it overwrites the values defined in the **Counter settings** window.

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

- 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

- You can save the current counter reading in a variable using the **FN 18: SYSREAD ID920 NR1** function.

  Further information: "Read system data with FN 18: SYSREAD", Page 629

- Client applications can change the counter reading using OPC UA and the NC.RemoteOperator role (#56-61 / #3-02-1*).

19.5.1 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

19.6 Table access with SQL statements

19.6.1 Fundamentals

Application

If you would like to access content in a table or manipulate the tables (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 633

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 variables 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 variables 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** establishes or removes connections between table columns and variables	Page 658
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	Page 659
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 661
SQL FETCH	**SQL FETCH** transfers the values to the bound variables	Page 665
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	Page 666
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	Page 668
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	Page 669
SQL INSERT	**SQL INSERT** creates a new table row	Page 671

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!

Even during simulation of an NC program, the control executes all SQL commands. Here, SQL commands may overwrite table values, for example, that also take effect in the **Program Run** operating mode. The overwritten values can lead to unexpected behavior or incorrect positioning during subsequent machining operations. There is a danger of collision.

▶ Skip SQL commands during simulations by using conditional jumps, for example

▶ 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.

19.6.2 Binding a variable to a table column with SQL BIND

Application

SQL BIND links a variable 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
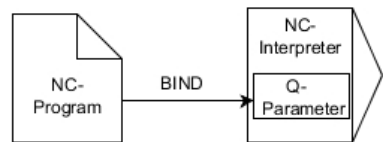
Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.

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 table access ▶ SQL BIND

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the **BIND** SQL command
Parameter	Variable to be bound
Name or **Parameter**	Table name and table column, separated by **.** or string parameter with definition Text or string parameter Only when binding a variable

Notes

- Enter the path of the table or a synonym as the table name.

  Further information: "Executing SQL statements with SQL EXECUTE", Page 661

- 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.

19.6.3 Reading out a table value with SQL SELECT

Application

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

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
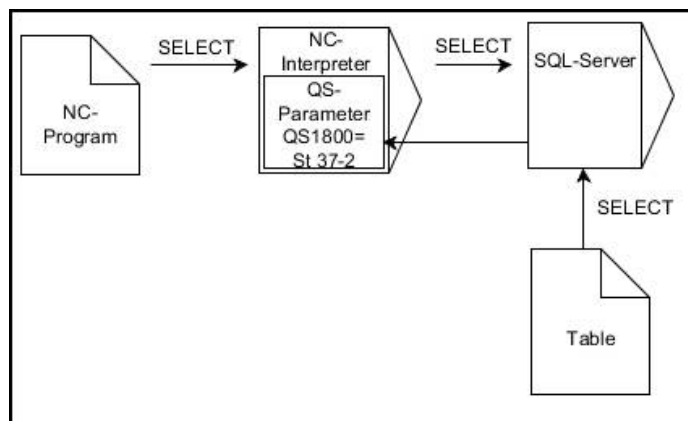
Black arrows and associated syntax show internal processes of **SQL SELECT**

With **SQL SELECT**, there is neither a transaction nor a binding between the table column and variable. 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 table access ▶ SQL SELECT

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL SELECT	Syntax initiator for the **SELECT** SQL command
Parameter	Variable in which the control stores the result
Name or **Parameter**	SQL statement or string 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 (variable after **:** in single quotation marks) Text, string parameter, or format string

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 as a variable as well. If you use numerical parameters for the comparison, the control rounds the defined value to an integer. If you use string parameters, the control uses the defined value.
- For the instructions within the SQL command, you can likewise use single or combined string parameters.
  Further information: "Concatenating values of string parameters", Page 645
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete 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	
*** - …**	

19.6.4 Executing SQL statements with SQL EXECUTE

Application

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

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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

Description of function
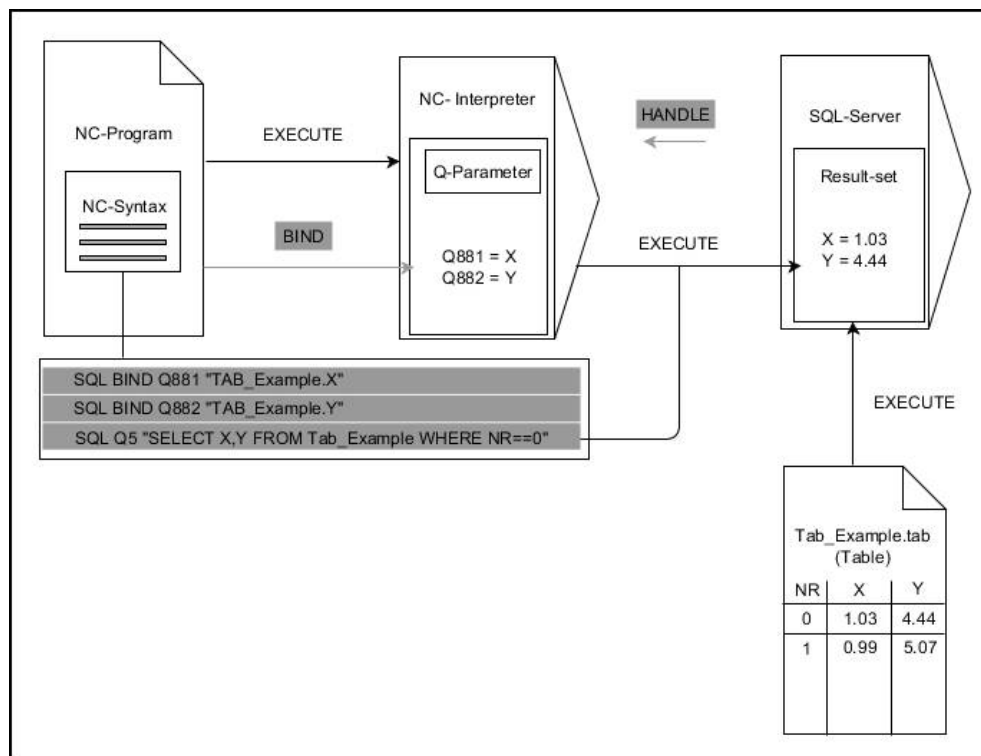
Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**. The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command.

The control provides the following SQL statements in the **SQL EXECUTE** command:

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete table
INSERT	Insert table rows
UPDATE	Update table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

SQL EXECUTE with the SQL SELECT instruction

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting with 0. The SQL commands **FETCH** and **UPDATE** use these row numbers (the **INDEX**).

SQL EXECUTE, in conjunction with the SQL instruction **SELECT**, selects the table values, transfers them to the **result set**, and always opens a transaction in the process. Unlike the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the **SELECT** instruction allows multiple columns and rows to be selected at the same time.

Enter the search criteria in the **SQL ... "SELECT...WHERE..."** function. You thereby restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

Enter the ordering criteria in the **SQL ... "SELECT...ORDER BY..."** function. This entry consists of the column designation and the keyword **ASC** for ascending or **DESC** for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If no rows meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

Conditions for WHERE entires

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Notes

- If you use the **SQL EXECUTE** NC function, the control will insert the **SQL** syntax element into the NC program only.
- You can also define synonyms for tables that have not yet been generated.
- The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.
- For the instructions within the SQL command, you can likewise use single or combined string parameters.
  Further information: "Concatenating values of string parameters", Page 645
- After the **WHERE** syntax element, you can define the comparison value as a variable as well. If you use numerical parameters for the comparison, the control rounds the defined value to an integer. If you use string parameters, the control uses the defined value.
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete 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"

Example: generating a table with CREATE TABLE

0 BEGIN PGM SQL_CREATE_TAB MM	
1 SQL Q10 "CREATE SYNONYM NEW FOR 'TNC:\table\NewTab.TAB'"	; Create synonym
2 SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM 'TNC:\prototype_for_NewTab.tab'"	; Create columns X, Y and Z of an existing table as a new table
3 END PGM SQL_CREATE_TAB MM	

0 BEGIN PGM SQL_CREATE_TABLE_QS MM	
1 DECLARE STRING QS1 = "CREATE TABLE "	
2 DECLARE STRING QS2 = "'TNC:\nc_prog\demo \Doku\NewTab.t' "	
3 DECLARE STRING QS3 = "AS SELECT "	
4 DECLARE STRING QS4 = "DL,R,DR,L "	
5 DECLARE STRING QS5 = "FROM "	
6 DECLARE STRING QS6 = "'TNC:\table\tool.t'"	
7 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
8 SQL Q1800 QS7	
9 END PGM SQL_CREATE_TABLE_QS MM	

19.6.5 Reading a line from a result set with SQL FETCH

Application

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored by the control in the bound variables. 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
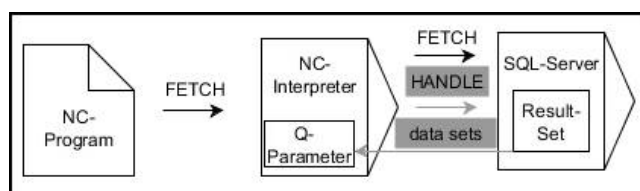
Black arrows and associated syntax indicate internal processes of **SQL FETCH**. The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command.

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
Parameter	Variable in which the control stores the result
HANDLE	Variable 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

19.6.6 Discarding changes to a transaction using SQL ROLLBACK

Application

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

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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

Description of function
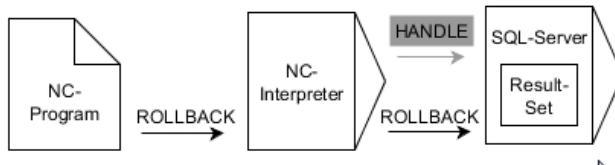
Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**. The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command.

The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
  - The control discards all changes and additions of the transaction
  - The control resets a lock set with **SELECT...FOR UPDATE**
  - The control completes the transaction (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (the control removes all of the other rows)
  - The control discards any changes and additions that may have been made in the non-specified rows
  - The control locks only those rows indexed with **SELECT...FOR UPDATE** (the control resets all of the other locks)
  - The specified (indexed) row is then the new Row 0 of the **result set**
  - The control does **not** complete the transaction (the **HANDLE** keeps its validity)
  - The transaction must be completed manually with **SQL ROLLBACK** or **SQL COMMIT** at a later time

Input

11 SQL ROLLBACK Q1 HANDLE Q5 INDEX 5	; Delete all rows of transaction **Q5** except row 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL ROLLBACK	Syntax initiator for the **ROLLBACK** SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction
INDEX	Row number within the **Result-set** as a number or variable that is retained Optional syntax element If not specified, the control discards all changes and additions to 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 ROLLBACK Q1 HANDLE Q5

19.6.7   Completing a transaction with SQL COMMIT

Application

SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. In this context, a lock that has been set with **SELECT...FOR UPDATE** resets the control.

Requirements

- Code number 555343
- Table exists
- Appropriate table name
  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function

The assigned **HANDLE** (operation) loses its validity.
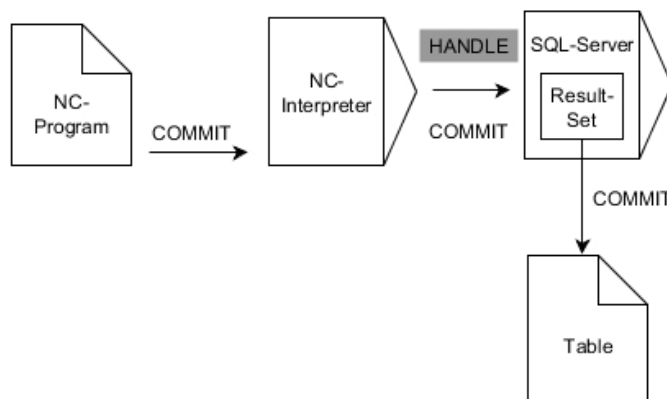
Black arrows and associated syntax indicate internal processes of **SQL COMMIT**.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL COMMIT Q1 HANDLE Q5	; Complete all rows of transaction **Q5** and update table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL COMMIT	Syntax initiator for the **COMMIT** SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable 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

19.6.8 Changing the row of a result set with SQL UPDATE

Application

SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound variables. 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
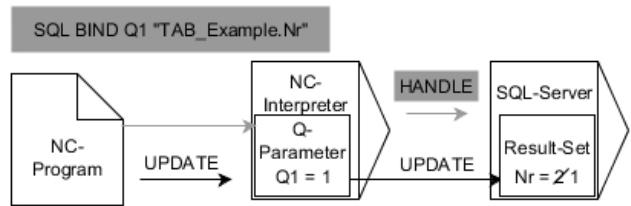
Black arrows and the associated syntax show internal **SQL UPDATE** processes. Gray arrows and the associated syntax are not directly associated with the **SQL UPDATE** command.

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
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction
INDEX	Row number within the **Result-set** as a number or variable
	Optional syntax element
	If not specified, the control accesses line 0.
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

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

Application

SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound variables. 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
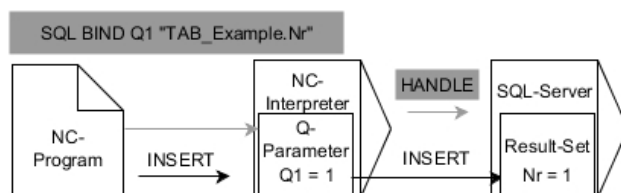
Black arrows and associated syntax indicate internal processes of **SQL INSERT**. The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command.

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
Parameter	Variable in which the control stores the result
HANDLE	Variable 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

19.6.10 Example

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

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

Use synonym

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

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

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

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

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

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

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

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

20

The Contour
graphics workspace

20.1 Fundamentals

Application

In the **Contour graphics** workspace, the control can draw contours directly during programming. You can also use graphical programming by drawing contours and exporting them as NC blocks. In addition, you can import contours from existing NC programs and edit them graphically.

Description of function

The **Contour graphics** workspace is available in the **Editor** operating mode.

Screen layout
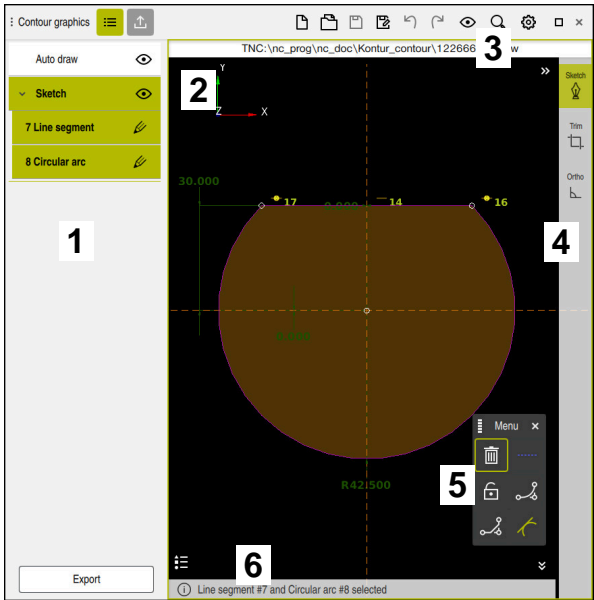
Screen layout of the **Contour graphics** workspace

The **Contour graphics** workspace contains the following areas:

1   Column **List**
2   Drawing area
3   Title bar
4   Toolbar
5   Drawing functions
6   Information bar

Controls and gestures in the Contour graphics workspace

Gestures

In addition to the gestures specifically available for graphical programming, you can also use various general gestures in graphical programming.

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

Further information: "Common gestures for the touchscreen", Page 84

Icons of the title bar

The **Contour graphics** workspace contains the following icons independent of the shown level:

Icon or shortcut	Meaning
	Open or close the **List** column
	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

Icon or shortcut	Meaning
⊕ (eye)	**Hide reference axes**
⊕ (eye)	**Show reference axes**
Q	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 682
▦	**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 682

Further information: "Icons on the control's user interface", Page 93

Icons and buttons of the List column
Depending on the selected level, the control displays the following icons in the **List** column:

Icon or button	Meaning
👁 (eye)	Show or hide the **Auto draw** and **Sketch** planes If you hide a plane, the control also hides the contour of this plane. You cannot edit the contour.
✏ (pencil)	Edit contour element Only in the **Sketch** plane
Move to " Sketch "	Shift contour elements to the **Sketch** plane Only in the **Auto draw** plane
Export	Export contour elements Only in the **Sketch** plane

Further information: "The List column", Page 681

Icons of the Sketch area

The control shows the following icons in the **Sketch** 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 not locked against editing. Select the icon to lock the element.
🔒	Unlock element	If this icon is displayed, the selected element is locked against editing. Select the icon to unlock 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 List column
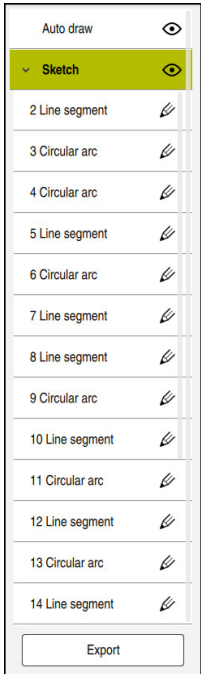
The **List** column with expanded **Sketching** area

In the **List** column, the control shows the **Auto draw** and **Sketching** areas.

Auto draw plane

In the **Auto draw** plane, the control draws the contour that you program in the **Program** workspace. For this purpose, the **Auto draw** switch must be active.

If you select the **Move to " Sketch "** button, the control shifts the contour from the **Auto draw** plane to the **Sketching** plane.

Further information: "Auto draw", Page 684

Sketching plane

The **Sketching** plane allows graphical programming of contours by drawing and exporting contours. Additionally, you can import contours from existing NC programs and edit them graphically.

In the **Sketching** plane, the control lists all contour elements of a drawn contour.

When you select the **Edit** icon, the control opens the element properties for this contour element. You cannot edit the element properties.

The control displays the following element properties, for example:

- Contour starting point
- Contour end point
- Length
- Restraints (e.g., horizontal)

In the **Sketching** plane you can export the contour.

Further information: "Exporting contours", Page 691

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** and **Diameter programming** settings are 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**
Diameter programming	Use a toggle switch to select whether sketched turning contours in the XZ and YZ planes are interpreted as radius or diameter dimensions during export (#50 / #4-03-1).
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

The 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 switch to 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.

Note

Define the **Contour settings** before auto-drawing or graphical programming.

Definitions

File type	Definition
H	NC program in Klartext format
TNCDRW	HEIDENHAIN contour file

20.2   Auto draw

Application

When the **Auto draw** function is active, the control draws the contour that you are programming in the **Contour graphics** workspace.

When you shift an auto-drawn contour into the **Sketch** plane, you can edit or export the contour.

Related topics

■ Drawing a new contour

Further information: "Creating a new contour", Page 687

■ Importing contours

Further information: "Importing contours for graphical programming", Page 688

Description of function
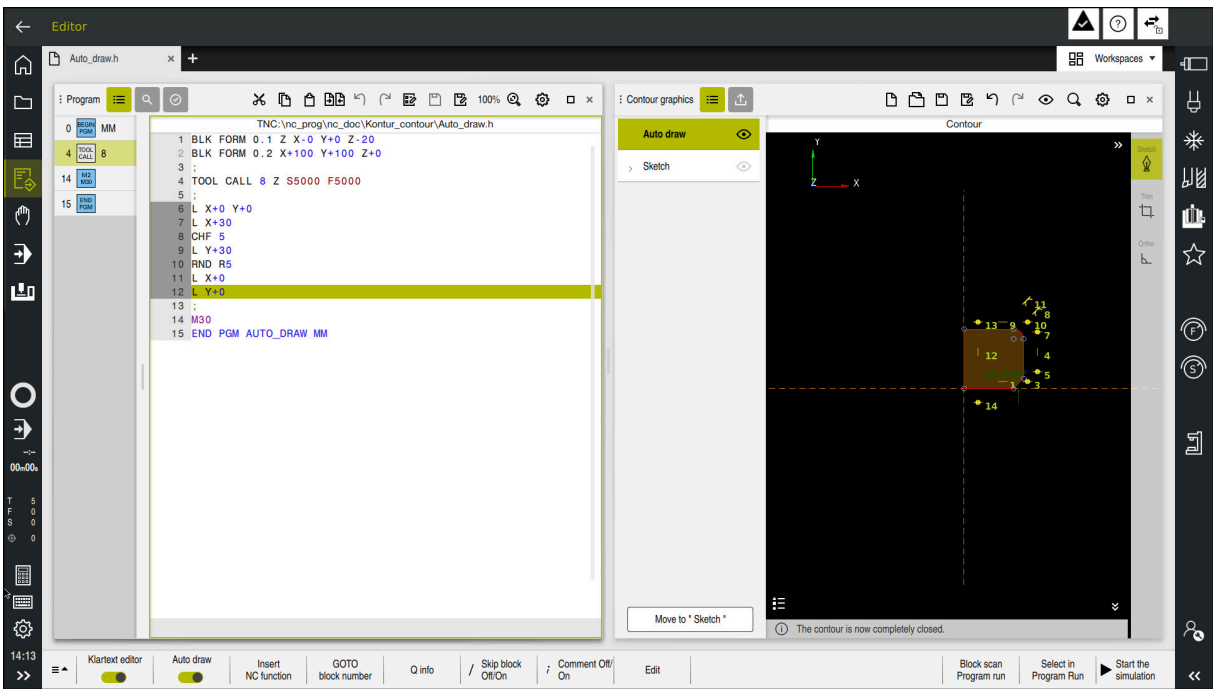
Auto-drawn contour from the NC program

The control highlights the auto-drawn NC block numbers in gray.

When the **Auto draw** function is active, you can program only path functions, except for approach and departure functions. For other NC functions the control will show an error message (e.g., for a cycle call).

The contour starting point must be clearly defined in the plane, which means, for example, it must include **X** and **Y** coordinates.

> **i** HEIDENHAIN recommends using the **Auto draw** function only during contour programming.

Auto draw a contour

To have a contour auto-drawn:

▶ Select the NC block from which the control starts auto draw

Auto draw ▶ Activate the **Auto draw** toggle switch

> The control opens the **Contour graphics** workspace, if necessary.

> The control highlights the NC block number of the selected NC block in gray.

▶ If applicable, open the **List** column

▶ Hide the **Sketch** plane

▶ Program the next contour point after the selected NC block

> The control auto-draws the contour element in the **Auto draw** plane of the **Contour graphics** workspace.

▶ Program further contour points as needed

> **i** If you mark several NC blocks and then activate the **Auto draw** switch, the control will draw the complete contour that has been marked.

Notes

- The control only auto-draws a coherent area with max. 200 NC blocks.
- To induce the control to draw a rounding **RND** or a chamfer **CHF**, you must also program the next contour element.

20.3   Graphical programming

20.3.1   Fundamentals of the graphical programming

Application

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

Description of function

You can use graphical programming independently via a separate tab or in the NC program. If you use graphical programming on its own tab, you cannot open any other workspaces of the **Editor** operating mode on this tab.

Possible elements and colors

The **Sketch** plane allows you to generate a contour as 2D sketch using different elements.

Further information: "First steps of graphical programming", Page 693

You have the choice of the following elements for drawing a contour:

- Line segment
- Arc
- Construction point
- Construction line
- Construction circle
- Chamfer
- Rounding arc

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.

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.

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

- Dimension each element immediately after drawing. If you do not dimension until the entire contour has been drawn, the contour may move unintentionally.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- You can assign constraints to the drawn elements. To avoid unnecessarily complicating the design, work only with necessary constraints.
  Further information: "Icons of the Sketch area", Page 679
- If you select elements of the contour, the control highlights the elements in the **List** column in green.

20.3.2 Importing contours for 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 the **Contour graphics** workspace, the control will output an error message.

Description of function
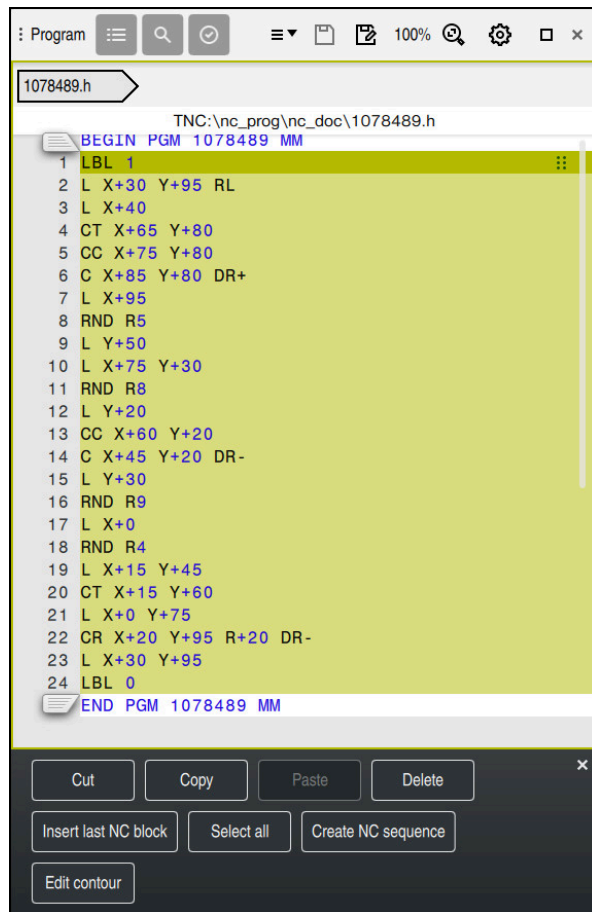
Contour to be imported from the NC program

In the **Contour graphics** workspace, all contours exclusively consist 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 235

■ NC blocks with polar coordinates

Further information: "Polar coordinates", Page 216

■ NC blocks with incremental inputs

Further information: "Incremental entries", Page 219

■ Free contour programming **FK**

Importing contours
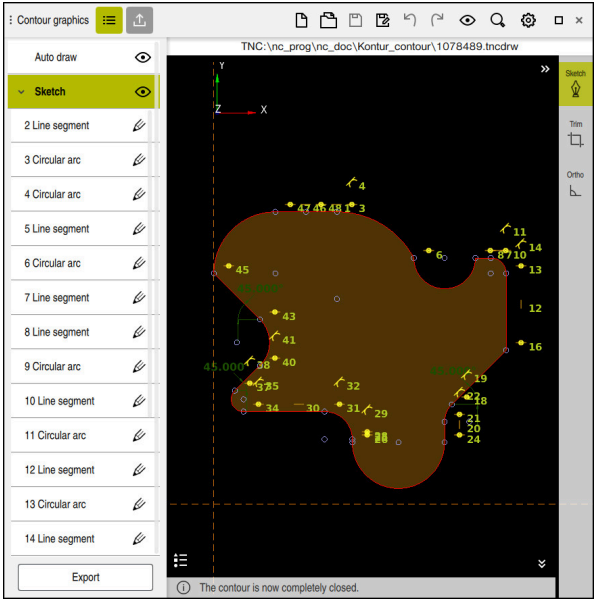
Imported contour

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
▶ Long press or right-click 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 **Sketch** plane of 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.

Notes

- You can also use the **Contour starting point** and **Contour end point** functions to use parts of the drawn elements and generate a contour from them.
- You can save drawn contours with the file type ***.tncdrw** to the control.
- When you export an imported or auto-drawn contour, the export does not include any technology data such as feed rates

20.3.3 Exporting contours

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 for graphical programming", Page 688

■ First steps in graphical programming

Further information: "First steps of graphical programming", Page 693

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 682
	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 use parts of the drawn elements and generate a contour from them.
- You can save drawn contours with the file type ***.tncdrw** to the control.
- When you export an imported or auto-drawn contour, the export does not include any technology data such as feed rates

20.3.4 First steps of graphical programming

Example assignment D1226664

Drawing a sample contour

To draw the displayed contour:

▶ Create a new contour
  Further information: "Creating a new contour", Page 687

▶ Configure **Contour settings**

> ℹ️ 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 682

 ▶ 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
 ▶ Select **Set the datum**
 > The control sets the center point of the circular arc to the.origin.
 ▶ 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.
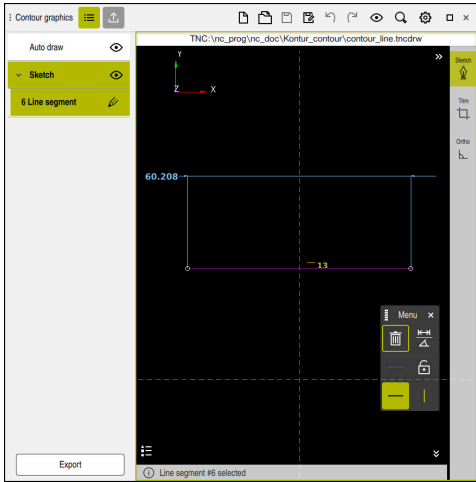
Line drawn
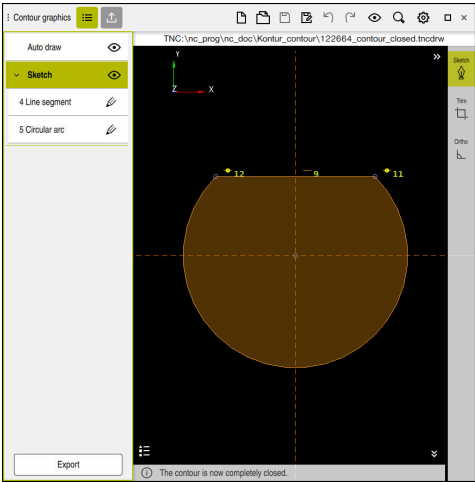
Closed contour
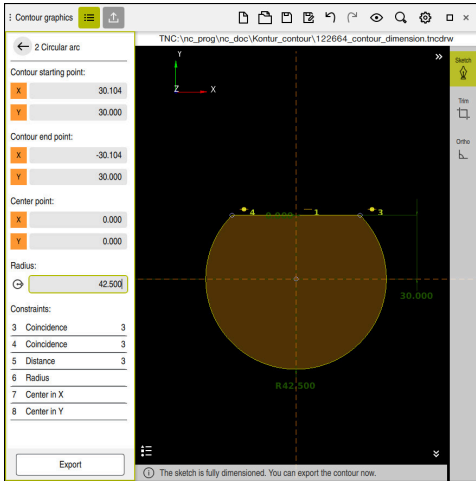
Dimensioned contour

Exporting a drawn contour

To export the drawn contour:

▶ Draw contour

⬆

▶ Select the **Export** column
> The control opens 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.
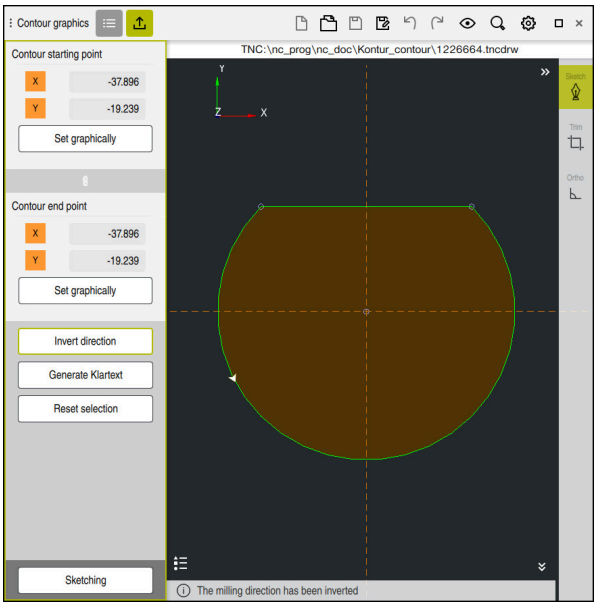
Selected contour elements in the **Export** column with defined **Milling direction**

21

ISO

21.1 Fundamentals

Application

The ISO 6983 standard defines a universal NC syntax.

Further information: "ISO example", Page 700

On the TNC7, you can program and execute NC programs using the supported ISO syntax elements.

Description of function

In connection with ISO programs, the TNC7 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 703

■ In addition to the standardized ISO syntax, you can program HEIDENHAIN-specific cycles as G functions.

Further information: "Cycles", Page 722

■ Coding in Klartext syntax allows you to use some NC functions in ISO programs.

Further information: "Klartext functions in ISO programming", Page 723

■ Testing of NC programs using Simulation mode

Further information: "The Simulation workspace", Page 761

■ 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	NC block numbers
N20	In the optional machine parameter **blockIncrement**
N30	(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 132

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	Syntax initiator
G90	Absolute or incremental input
	Further information: "Absolute and incremental input", Page 703
X+10 Y+0	Coordinates
	Further information: "Fundamentals of coordinate definitions", Page 216
G41	Tool radius compensation
	Further information: "Tool radius compensation", Page 713
F3000	Feed rate
	Further information: "Feed rate", Page 705
M3	Miscellaneous functions (M functions)
	Further information: "Miscellaneous Functions", Page 553

ISO example

Example task 1338459
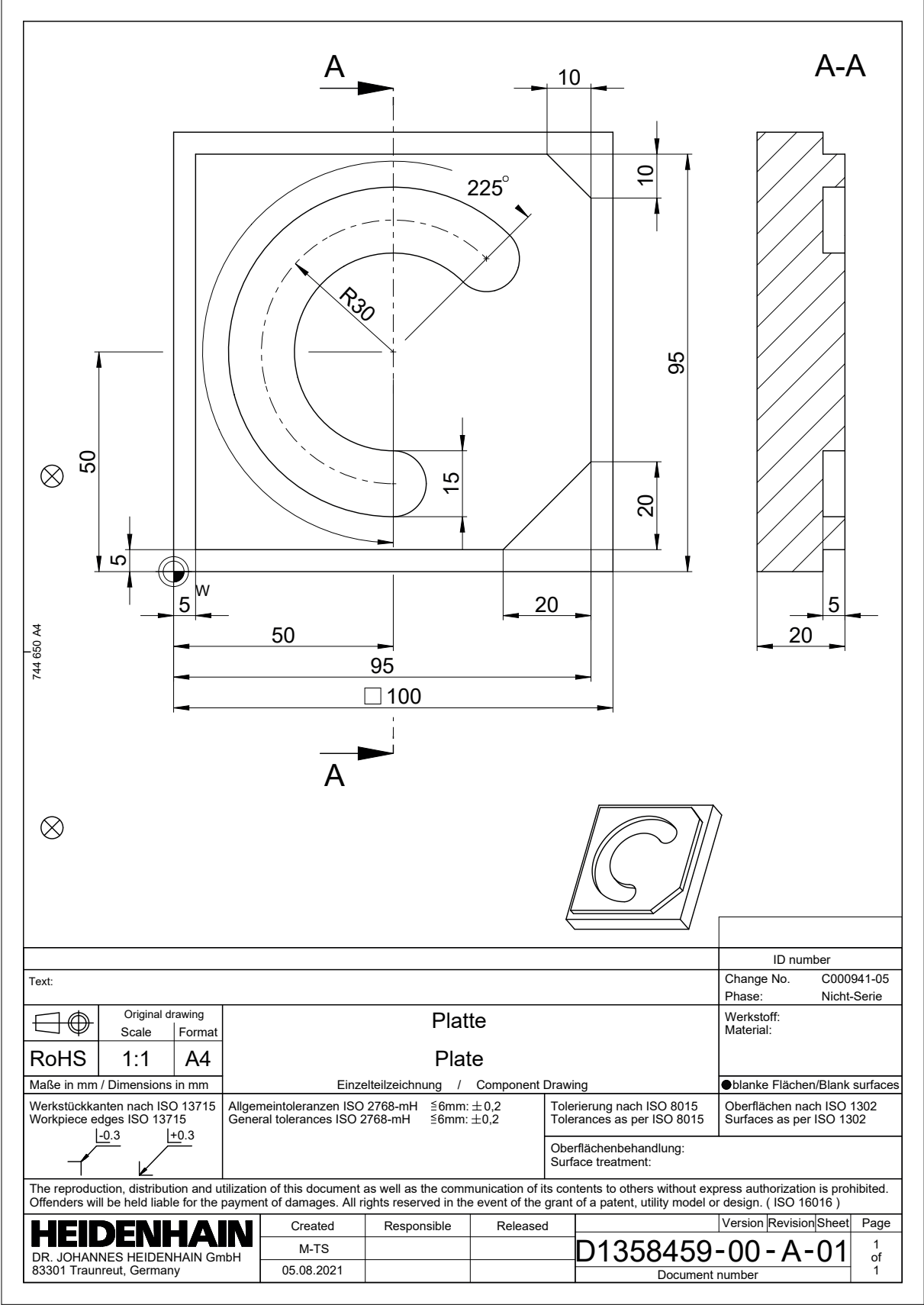
Example solution 1338459

% 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 148
- 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 711
  Further information: "The Contour graphics workspace ", Page 675
- 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 POLARKIN", Page 532
- If the **ISO editor** toggle switch is active, you can search for identical syntax elements in different NC blocks.
  Further information: "Searching for identical syntax elements in different NC blocks", Page 146

21.2 ISO syntax

Keys

You can use the following keys to insert ISO syntax:

Key	ISO syntax	Further information
TOOL CALL	Tool call **T**	Page 704
TOOL DEF	Tool definition **G99**	Page 705
L	Straight line **G01**	Page 706
CHF	Chamfer **G24**	Page 706
RND	Rounding arc **G25**	Page 707
CC	Circular arc **G02**	Page 708
C	Circular arc **G03**	Page 708
CR	Circular arc **G05**	Page 708
CT	Tangential arc **G06**	Page 709
LBL SET	Label **G98**	Page 710
LBL CALL	Subprogram call and program-section repeat **L**	Page 711 Page 711
STOP	Stop in the NC program **G38**	Page 713

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 186

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 704

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 190

- Rotationally symmetric workpiece blank with **BLK FORM ROTATION**

  Further information: "Rotationally symmetric workpiece blank with BLK FORM ROTATION", Page 191

- STL file as workpiece blank with **BLK FORM FILE**

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 193

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: "Using TOOL CALL to call a tool", Page 205

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 704

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 210

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 211

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 using TOOL DEF", Page 213

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 using TOOL DEF", Page 213

Path functions

Straight line

Cartesian coordinates

With the **G00** and **G01** NC functions, you program a straight movement in rapid traverse or with a machining feed rate in any desired direction.

N110 G00 Z+100 M3	; Straight line at rapid traverse
N120 G01 X+20 Y-15 F200	; Straight line at machining feed rate

If the feed rate was programmed using a numerical value, it is active only up to the NC block in which a new feed rate is programmed. **G00** is active only for the NC block in which it was programmed. When the NC block programmed with **G00** has been executed, the feed rate programmed most recently with a numerical value becomes active again.

> **i** Make sure to program rapid traverse movements exclusively with the **G00** NC function instead of very high numerical values. This is the only way to ensure that rapid traverse is active on a block-by-block basis and that you can control rapid traverse independently of the machining feed rate.

G00 and **G01** correspond to the **L** Klartext syntax with **FMAX** and **F**.

Further information: "Straight line L", Page 224

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 243

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 226

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 227

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 229

> 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 239

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 241

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 230

> When you program a radius **R**, there is no need to define a circle center.
>
> **Further information:** "Circular arc with a defined radius", Page 709

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 245

Circular arc 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 232

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 235

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 247

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 256

You can also use polar coordinates when programming the **APPR** and **DEP** NC functions.

Further information: "Approach and departure functions with polar coordinates", Page 270

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 282

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 282

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 283

> **i** In order to define a certain number of desired repetitions (e.g., **L1.3**), program a program-section repeat.
> **Further information:** "Program-section repeat", Page 711

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 285

Selection functions

Further information: "Selection functions", Page 300

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: "Calling an NC program with CALL PGM", Page 300

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 a datum table in the NC program", Page 332

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: "The Contour graphics workspace ", Page 675

%: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: "Calling an NC program with CALL PGM", Page 300

Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 302

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|G00 corresponds to the Klartext syntax **CYCL CALL POS** with **FMAX**	The control calls the most recently programmed machining cycle at the position you defined in the NC block with **G79|G00**. The control moves to the defined position at rapid traverse.

Syntax	Meaning
G79|G01 corresponds to the Klartext syntax **CYCL CALL POS** with **F**	The control calls the most recently programmed machining cycle at the position you defined in the NC block with **G79|G01**. The control moves to the defined position at the machining feed rate.
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 412

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 553

G38 corresponds to the **STOP** Klartext syntax.

Further information: "Miscellaneous functions M and the STOP function ", Page 554

Programming variables

The control provides the following options for programming variables in ISO programs:

Function group	Further information
Basic arithmetic operations	Page 715
Trigonometric functions	Page 716
Circle calculations	Page 717
Jump commands	Page 718
Special functions	Page 720
String functions	Corresponds to the Klartext syntax Page 642
Counters	Corresponds to the Klartext syntax Page 653
Calculations using formulas	Corresponds to the Klartext syntax Page 636
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: "Programming with variables", Page 597

> **i** Not all NC functions for programming variables are available in ISO programs (e.g., accessing tables with SQL statements).
> **Further information:** "Table access with SQL statements", Page 655

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 613

> **i** HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
> **Further information:** "Formulas in the NC program", Page 636

Trigonometric functions

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

The control provides the following functions:

Syntax	Meaning
D06	Sine
	Calculate and assign the sine of an angle in degrees
D07	Cosine
	Calculate and assign the cosine of an angle in degrees
D08	Root of the sum of squares
	Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
D13	Angle
	Calculate and assign the angle from the opposite side and the adjacent side using arctan or from the sine and cosine of the angle (0 < angle < 360°)

N110 D06 Q20 P01 -Q5	; Sine, Q20 = sin(−Q5)
N110 D07 Q21 P01 -Q5	; Cosine, Q21 = cos(−Q5)
N110 D08 Q10 P01 +5 P02 +4	; Root of the sum of squares, Q10 = $\sqrt{(5^2+4^2)}$
N110 D13 Q20 P01 +10 P02 -Q1	; Angle, Q20 = arctan(25/−Q1)

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Trigonometric functions folder", Page 616

ℹ️ HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.
Further information: "Formulas in the NC program", Page 636

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 617

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 619

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 633
D27	Write to a freely definable table
	Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 633
D28	Read from a freely definable table
	Further information: "Reading a freely definable table with FN 28: TABREAD", Page 635

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 621
D16	Output formatted texts
	Further information: "Outputting text formatted with FN 16: F-PRINT", Page 622
D18	Read system data
	Further information: "Read system data with FN 18: SYSREAD", Page 629
D19	Transfer values to the PLC
	Further information: "Special functions defining the machine behavior", Page 848
D20	Synchronize NC and PLC
	Further information: "Special functions defining the machine behavior", Page 848
D29	Transfer values to the PLC
	Further information: "Special functions defining the machine behavior", Page 848
D37	Create user-defined cycles
	Further information: "Special functions defining the machine behavior", Page 848
D38	Send information from the NC program
	Further information: "Sending information from the NC program with FN 38: SEND", Page 631

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

21.3   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 353

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 711

- If you do not use **%:TAB:**, you have to activate the datum table manually.

  Further information: "Activating the datum table manually", Page 331

21.4 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**

■ Programs that use normal vectors
Further information: "CAM-generated NC programs", Page 538

■ Program defaults for cycles with **GLOBAL DEF**
Further information: User's Manual for Machining Cycles

■ Table access with SQL statements
Further information: "Table access with SQL statements", Page 655

22

User aids

22.1　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 52

- User's Manual as the **TNCguide** integrated product aid

  Further information: "User's Manual as integrated product aid: TNCguide", Page 51

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 134

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 139
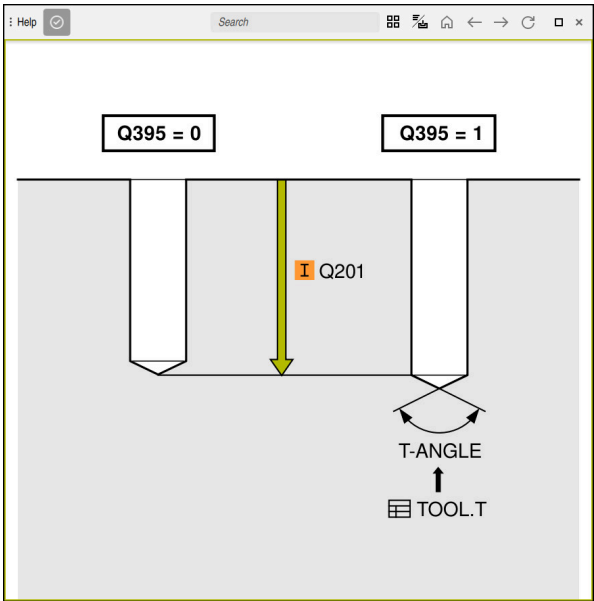
The **Help** workspace with a help graphic of a cycle parameter

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 51
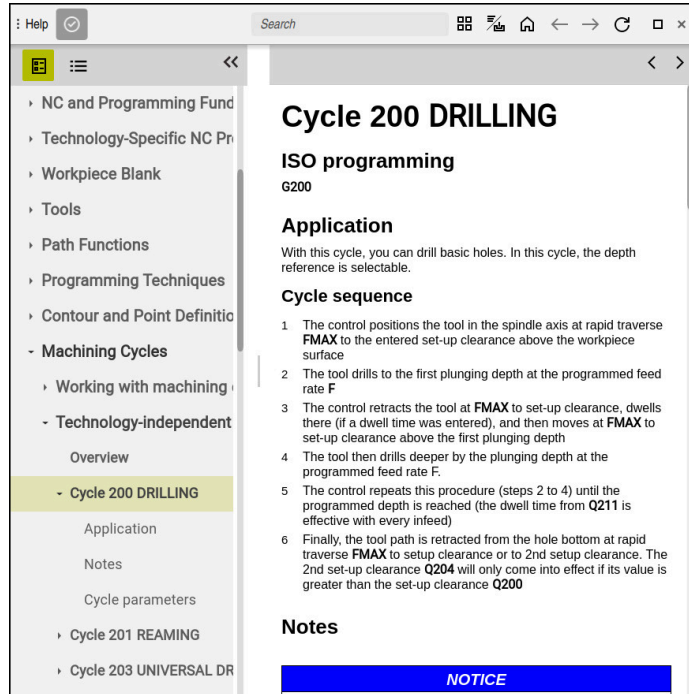
The **Help** workspace with **TNCguide** open

Icons

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

Icon	Meaning
⊘	Open or close the **Search results** column
	Further information: "Searching in TNCguide", Page 54
⊞	**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 53
≡⌐	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 52

Icon	Meaning
← →	**Navigate** Navigate between the contents opened recently
↻	**Refresh**

TNCguide has additional icons.

Further information: "User's Manual as integrated product aid: TNCguide", Page 51

22.2 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.

The **+**, **-**, *****, **/**, **(** and **)** keys permit calculations concerning numerical values in input fields and table rows.

NC input areas
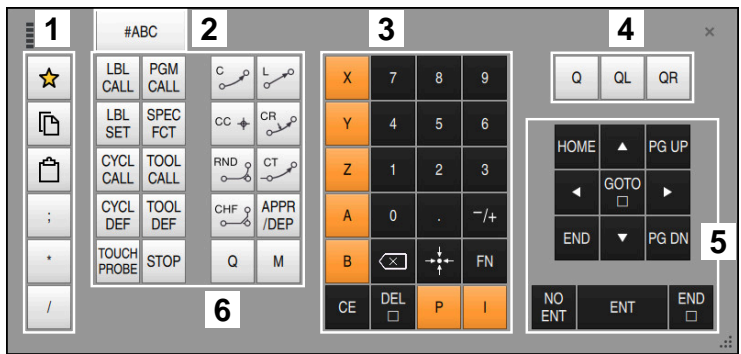
Virtual keyboard in NC input mode

NC input mode contains the following areas:

1 File functions
- Define favorites
- Copy
- Paste
- Add comment
- Add structure item
- Hide NC block
2 Switch to text input
3 Axis keys and numerical input
4 Q parameters
5 Navigation and dialog keys
6 NC functions

> **i** If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
> - **Q**
> - **QL**
> - **QR**

Text input areas
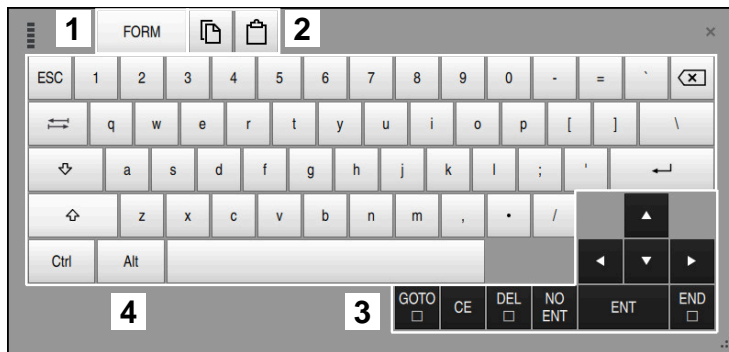
Virtual keyboard in text input mode

Text input mode contains the following areas:

1   Switch to formula input
2   Copying and pasting
3   Navigation and dialog keys
4   Input

Formula input areas
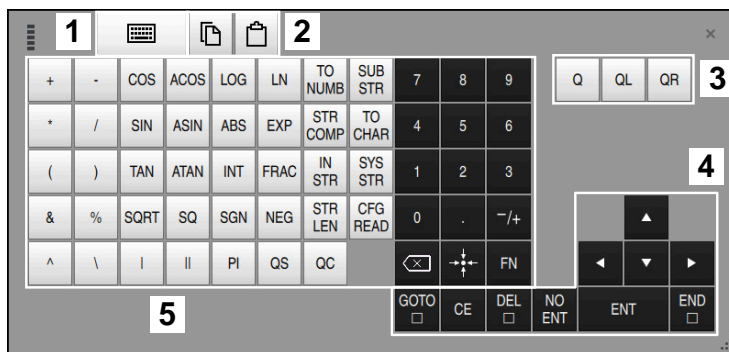
Virtual keyboard in formula input mode

Formula input mode contains the following areas:

1   Switch to NC input
2   Copying and pasting
3   Q parameters
4   Navigation and dialog keys
5   Input

22.2.1   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.

22.3   GOTO function

Application

Use the **GOTO** function for navigating rapidly and purposefully within NC programs, text files or tables.

Related topics

■ Selecting an NC block for program run with **Block scan**

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

Description of function

The control offers the **GOTO** function in the following areas of the control:

■ The **Program** workspace

Further information: "The Program workspace", Page 136

■ The **Text editor** workspace

Further information: "The Text editor workspace", Page 459

■ The **Table** workspace

Further information: "The Table workspace", Page 813

The **GOTO block number** button allows you in the **Program** workspace to define an NC block selected by the control. 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.

The **GOTO record** button allows you in the **Table** and **Text editor** workspaces to define a row selected by the control.

22.3.1   Selecting an NC block or row with GOTO

To select an NC block or a row:

▶ Select **GOTO**

> The control opens the **GOTO jump instruction** window.

▶ Enter the block or row number

▶ Press **OK**

> The control selects the defined NC block or the defined row.

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

- You can also use the **GOTO** function with the **CTRL + G** shortcut.
- If the control in the action bar shows an icon for selection, you can open the selection window with **GOTO**.
- If, for example, the correct row number is unknown, the **Program** and **Text editor** workspaces allow you in the **GOTO jump instruction** window to open the **Search** column, using an icon.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

22.4 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 be edited only in the Text editor mode or in the **Form** column.

Further information: "Using the Program workspace", Page 144

22.4.1 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

22.4.2 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

22.4.3 Commenting an NC block out or in

Use the **Comment out/in** button to define an existing NC block as a comment or to change a comment back to an NC block.

To comment an existing NC block in or out:
- ▶ Select the desired NC block

; Comment Off/On	▶ Select **Comment Off/On**
	> The control inserts a **;** character at the beginning of the block.
	> If the NC block is already defined as a comment, the control removes the **;** character.

22.5 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 during 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** switch in the **Program Run** operating mode or in the **MDI** application, the control skips this NC block during program run.

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

22.5.1 Hiding or showing NC blocks

To hide or show an NC block:

▶ Select the desired NC block

/ Skip block Off/On	▶ Select **Skip block Off/On**
	> The control adds a **/** character before the NC block.
	> If the NC block is already hidden, the control removes the **/** character.

> **ℹ** Instead of the button, you can also use the following keys:
> ■ Hiding oder showing with the **/** key
> ■ Showing with the **BACKSPACE** key
> **Further information:** "Keycaps for alphabetic keyboard", Page 85

22.6   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 737

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 737

22.6.1   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

22.7 The Structure column in the Program workspace
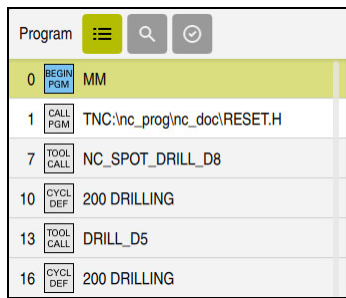
Application

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 139

■ Inserting structure items manually

Further information: "Structuring of NC programs", Page 736

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 139

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
FCT MODE	**FUNCTION MODE**	▪ Selected machining mode (possibilities: **MILL**, **TURN**, 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	**END PGM**	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.

> **i** 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 734

22.7.1 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.

22.7.2 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 747
▶ Select **Mark**
> The control displays check boxes next to the structuring items in the **Structure** column.
> The control marks the NC block in the NC program.
▶ Enable additional check boxes, if required
> The control marks all structuring items between the two selected structuring items as well as the associated NC blocks.

> **i** Instead of the context menu, you can use the **CTRL + SPACE** shortcut.

Notes

- In the case of long NC programs, generating the structure view may take longer than loading the NC program itself. Even if the structure view has not been fully generated, you can already work in the loaded NC program.
- You can navigate within the **Structure** column using the up and down arrow keys.
- The control shows called NC programs in the structure with a white background. If you double-tap or click on such a structure element, the control opens the NC program if necessary in a new tab. If the NC program is open, the control switches to the corresponding tab.

22.8 The Search column in the Program and Text editor workspaces

Application

In the **Search** column, you can search files for any character strings, such as individual syntax elements in the NC program. 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 identical syntax elements in different NC blocks", Page 146
- The **Search** column in the **Table** workspace
  Further information: "The Search column in the Table workspace", Page 818
- The **Program** workspace
  Further information: "The Program workspace", Page 136
- The **Text editor** workspace
  Further information: "The Text editor workspace", Page 459

Description of function
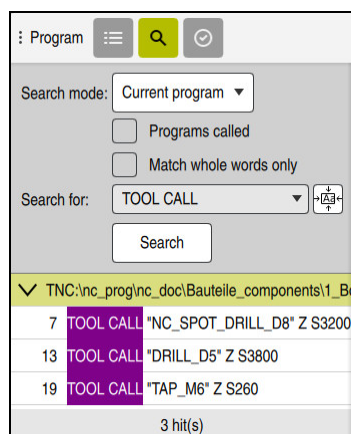
The **Search** column in the **Program** workspace

The **Search** column can be selected in the **Program** and **Text editor** workspaces. In the **MDI** application you search only in 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:

Icon or button	Meaning
Search mode:	■ **Current program** or **Current file** Search current file ■ **Opened programs** or **Opened files** Search all opened files of the same type (e.g., ***.h** or ***.txt**) ■ **Search and replace** Search for strings and replace them with new strings, such as syntax elements **Further information:** "Search and replace mode", Page 743
Programs called	Search all called NC programs of the current NC program Only in the **Current program** mode
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**.
Search for:	In the input area, you define the search term. If you have not yet entered any characters, the control suggests the last six search terms for selection. The search is not case-sensitive.
→Aa←	The control loads the word or syntax element at the current cursor position into the input area. If the selected NC block is not edited, the control accepts the syntax initiator.
Search	Start search

The control shows the following information about the results:
■ File paths
■ Row numbers or NC block numbers
■ Complete row contents
■ Number of results

The control groups the results according to files. If you select a result, the control positions the cursor in the corresponding row.

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 in the NC programs 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:

Buttons	Meaning
Search backward	The control searches the file from bottom to top.
Wrap around	The control searches the entire file, wrapping around the start and end.
Find next	The control searches the file for the search term. The control marks the next result in the file.
Replace	The control performs a syntax check, if required, and automatically replaces the marked content 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, if required, 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, if required, and automatically replaces all found results with the contents of the **Replace with:** field.

22.8.1 Search for and replace syntax elements

To search for and replace syntax elements in the NC program:

⬚ ▸ Select an operating mode (e.g., **Editor**)

     ▸ Select the desired NC program

     > The control opens the selected NC program in the **Program** workspace.

🔍 ▸ Open the **Search** column

     ▸ In the **Search mode:** field, select the **Search and replace** function

     > The control displays the **Search for:** and **Replace with:** fields.

     ▸ In the **Search for:** field, enter the search content (e.g., **M4**)

     ▸ In the **Replace with:** field, enter the desired content (e.g., **M3**)

[Find next] ▸ Select **Find next**

     > The control closes previously called NC programs, if any had been called, and highlights the first result in the main program in purple.

[Replace] ▸ Select **Replace**

     > The control performs a syntax check and replaces the content if the check is successful.

Notes

- The search results are retained until you shut down the control or search again.
- If you double-tap or click on a search result in a called NC program, the control opens the NC program (on a new tab if not already open). If the NC program is already open, the control switches to the corresponding tab.
- If you have not entered a value for **Replace with:**, the control deletes the search value.

22.9 Program comparison

Application

Use the **Program comparison** function to determine differences between two NC programs. You can transfer the differences 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 132

Description of function
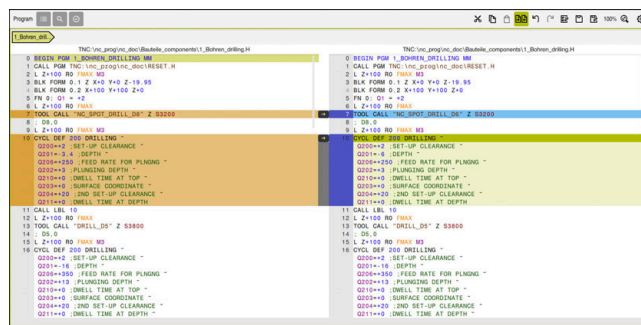
Program comparison of two NC programs

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.

22.9.1   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 747

22.10 Context menu

Application

With a long-press gesture or by right-clicking with the mouse, the control opens a context menu for the selected element, such as an NC block or file. Use the various functions of the context menu to run commands that affect the currently selected element(s).

Description of function

The functions available in the context menu depend on the selected element as well as the selected operating mode.

General
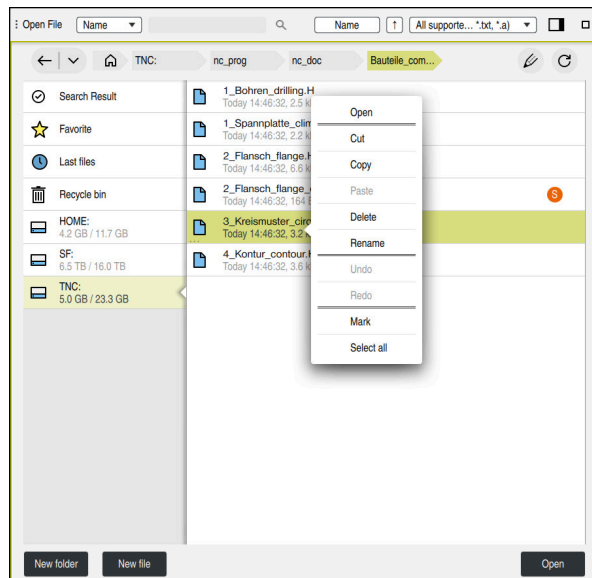
Context menu in the **Open File** workspace

Depending on the selected workspace and operating mode, the context menu provides the following functions:

- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Undo**
- **Redo**
- **Mark**
- **Select all**

> **i** If you select the **Mark** or **Select all** functions, the control opens the action bar. The action bar displays all functions that are currently available for selection from the context menu.

As an alternative to the context menu, you can use keyboard shortcuts:

Further information: "Icons on the control's user interface", Page 93

Key or keyboard shortcut	Meaning
CTRL + SPACE	Mark the selected line
SHIFT + UP	▪ Mark the selected line ▪ Additionally mark the line directly above ▪ During the editing, mark all NC blocks up to the preceding same syntax element
SHIFT + DOWN	▪ Mark the selected line ▪ Additionally mark the line directly below ▪ During the editing, mark all NC blocks up to the next same syntax element
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**
- **Open in text editor**
- **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 747

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 808

Context menu in the Job list workspace
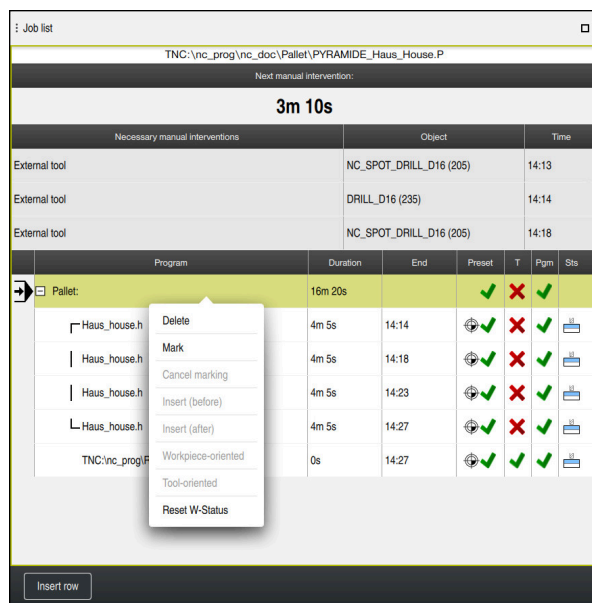
Context menu in the **Job list** workspace

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 789

Context menu in the Program workspace
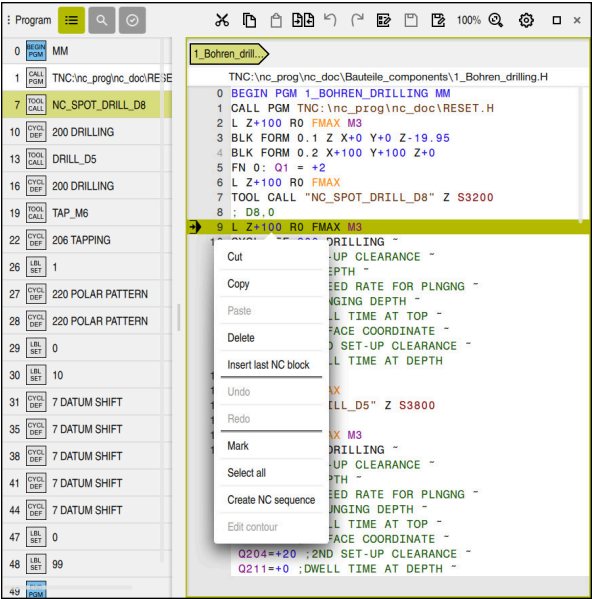
Context menu in the **Program** workspace of the **Editor** operating mode

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 305

- **Edit contour**

    Only in the **Editor** operating mode

    Further information: "Importing contours for graphical programming", Page 688

The following functions in the context menu of the **Program** workspace are available only if you select a value of an NC block:

- **Select value**
- **Replace value**

  Add values from the clipboard in the NC block such as values from the calculator

Further information: "The Program workspace", Page 136

> ℹ️ The **Select value** and **Replace value** functions are only available in the **Editor** operating mode and in the **MDI** application.
>
> The **Replace value** function also works if no value has been defined yet for the syntax element. In this case the otherwise necessary marking of the value to be replaced is omitted.

Further information: "Calculator", Page 754

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 148

Context menu in the Document workspace
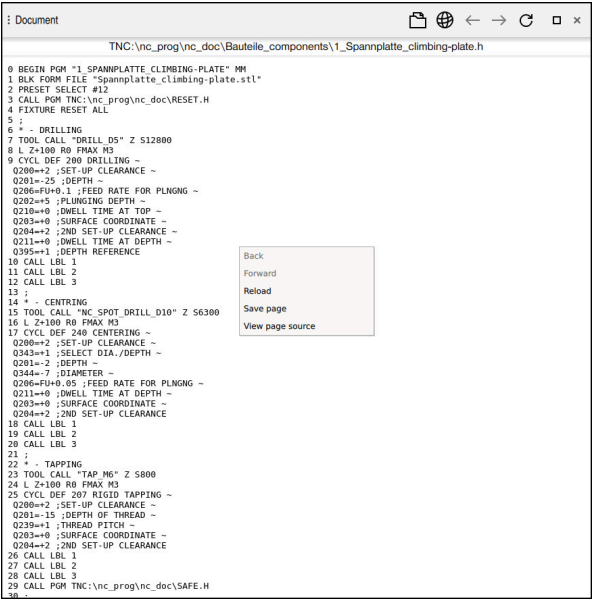
Context menu in the **Document** workspace

In the **Document** workspace, the context menu offers the following additional functions:

Function	Meaning	File types
Back	Navigate backwards between the opened files	▪ All supported file types
Forward	Navigate forwards between the opened files	▪ All supported file types
Reload	Reload the opened file	▪ All supported file types
Save Page	Currently no function	▪ All supported file types
View page source	Currently no function	▪ PDF ▪ HTML ▪ Text files
Save image	Currently no function	▪ Image files
Copy image	Currently no function	▪ Image files
Copy image address	Copy opened file to clipboard	▪ Image files
Loop	Play video in a loop	▪ Video files
Save media	Currently no function	▪ Video files
Copy media address	Copy opened file to clipboard	▪ Video files

Further information: "The Document workspace", Page 454

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**

> Instead of **Insert** you can also use the **CTRL + N** shortcut in the configuration editor.
> **Further information:** "General", Page 747

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

22.11 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 and load current axis positions.

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.

> **i** The control additionally offers the following functions of the calculator:
> - The **P** key corresponds to **PI**
> - The **RETURN** or **ENT** key corresponds to **=**
>
> **Further information:** "Operating elements of the keyboard unit", Page 84

22.11.1 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.

22.11.2 Actual position capture

To load the actual position of an axis into the calculator:

▶ Select **actual position capture**
> The control opens the position display in the calculator.
▶ If required, select the mode for the position display such as **Actual pos. (ACT)**
▶ Select the desired value
> The control transfers the value to the calculator.
▶ Select **actual position capture**
> The control closes the position display.

> **i** The calculator shows the same mode of the position display as the status overview of the TNC bar. If you change the mode, the control synchronizes the two position displays.
>
> The position display mode can be selected independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).
>
> **Further information:** User's Manual for Setup and Program Run

22.11.3 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.

22.11.4 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.

22.12 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 757

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 835
- Tool cutting material in table **TMAT.tab**
  Further information: "Table for tool materials TMAT.tab", Page 835
- Combination of workpiece material and cutting material in the cutting data table ***.cut** or in the diameter-dependent cutting data table ***.cutd**

> **i** 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 836
>
> 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 837

- 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

22.12.1 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.

22.12.2 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.

Notes

■ Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
■ You cannot perform any cutting data calculation in turning mode (#50 / #4-03-1) with the cutting data calculator because the feed rate and spindle speed data in turning mode are different from those in milling mode.

In turning operations the feed rates are often defined in millimeters per revolution (mm/1) (**M136**), whereas the cutting data calculator always calculates feed rates in millimeters per minute (mm/min). Furthermore, the radius in the cutting data calculator is referenced to the tool; turning operations, however, require the workpiece diameter.

22.13 Issuing messages with FUNCTION REPORT

Application

With the **FUNCTION REPORT** NC function, the controls issues notifications under program control. You can define the notification text yourself. If the machine manufacturer or another provider has saved notifications as a PO file, you can also output these notifications.

Related topics

- Issuing error messages with **FN 14: ERROR**

  Further information: "Output error messages with FN 14: ERROR", Page 621

- Message menu

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

Description of function

When the NC function **FUNCTION REPORT** is executed, the control displays a message and triggers a reaction.

With the NC function you can program the following reactions:

Syntax element	Message type	Reaction
WARNING	Warning	NC program continues to run
STOP	Error	Program run or simulation stops
ERROR	Error	Program run or simulation is aborted

Input

Example

11 FUNCTION REPORT WARNING DOMAIN: "PO-File" ID: "EXAMPLE"	; Issue message from the PO file "EXAMPLE"

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Functions ▶ FUNCTION REPORT

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION REPORT	Syntax initiator for issuing a message
ERROR, **STOP** or **WARNING**	Reaction
TEXT: or **DOMAIN:**	Enter message text yourself or select it from a PO file
Name or **Parameter**	Text or PO file When selecting **TEXT:** Text, string parameter, or format string When selecting **DOMAIN:**, text entry as well as selecting from a selection window are possible
ID:	ID of the text from the PO file Only if **DOMAIN:** has been selected

Note

Refer to your machine manual.

Only existing PO files can be output which have been stored by the machine manufacturer or a third party provider. In order to output the PO files, the machine manufacturer must provide the IDs of the texts.

The PO file must contain the following information:

Meaning	Input
Message text	msgid "EXAMPLE" msgstr "Message text"
Cause	msgid "EXAMPLE_|UR|" msgstr "Cause"
Corrective action	msgid "EXAMPLE_|BE|" msgstr "Corrective action"

23

The Simulation workspace

23.1 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 186

Description of function

In the **Editor** operating mode, the **Simulation** workspace can be open for only one NC program at a time. With this NC program the control displays the **Control-in-operation** icon next to the program name. 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 768

If the simulation of another NC program is currently running, the control shows a window with the name of this NC program above the function bar. If you double-tap or click this window, the control switches from the active tab to the NC program currently being simulated.

The functions available in the simulation depend on the following settings:

- Selected model type, 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 764
🧰	Open or close the **Workpiece options** column
	Further information: "The Workpiece options column", Page 766
⬚	Open or close the **Pre-defined views** selection menu
	Further information: "Pre-defined views", Page 773
📝	**Save as**
	Export simulated workpiece as STL file
	Further information: "Exporting a simulated workpiece as STL file", Page 774
⚙	Open or close the **Simulation settings** window
	Further information: "The Simulation settings window", Page 768
👁	Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
	DCM active
👁	DCM inactive
	Further information: "The Visualization options column", Page 764
👁	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 479
✓++	**Advanced checks** active
	Further information: "The Visualization options column", Page 764
⬚	Model quality
	Further information: "The Simulation settings window", Page 768
T 0	Number or name of the current tool
	ℹ The display depends on the workspace size.
00:00:00	Current run time of the program in hh:mm:ss format

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 471	■ The **Editor** operating mode ■ Simulation reset or not started yet

Icon or toggle switch	Meaning	Requirements
Advanced checks	If you activate the **Advanced checks** toggle switch, the control provides the following checks: ■ **Rapid-traverse cut** ■ **Collision between workpiece and tool** ■ **Collision between workpiece and machine** ■ **Fixture collision** **Further information:** "Advanced checks in the simulation", Page 481	■ 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 offers 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 735
	 If the toggle switch is active, then the control dims the NC blocks to be skipped.
	 Further information: "Appearance of the NC program", Page 138
	
	 ■ **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 555
	 If the toggle switch is inactive, then the control dims the **M1** syntax element.
	 Further information: "Appearance of the NC program", Page 138
		■ 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 between the measured surface and the finished part, using only the **3D** model type. **Further information:** "Measuring function", Page 776	■ The **Workpiece** mode ■ The **2.5D** or **3D** model type
Cutout view	Use this function to cut through the simulated workpiece along a plane. **Further information:** "Cutout view in the simulation", Page 778	■ The **Workpiece** mode ■ The **Editor** operating mode ■ The **2.5D** model type
Highlight workpiece edges	Use this function to highlight the edges of the simulated workpiece.	■ The **Workpiece** mode ■ The **2.5D** model type
Workpiece blank frame	Use this function to show the outside lines of the workpiece blank.	■ The **Workpiece** mode ■ The **Editor** operating mode ■ The **2.5D** model type
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 778	
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 768	■ 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 780
	 ■ **Monitoring**
	 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 498
	   Further information: User's Manual for Machining Cycles
	   ■ Process heatmap with **SECTION MONITORING** (#168 / #5-01-1)
	   Further information: "Process monitoring (#168 / #5-01-1)", Page 501
		■ The **2.5D** model type
		 ■ **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 **2.5D** model type
Reset the tool paths	Use this function to reset the simulated tool paths.	■ The **Workpiece** mode ■ The **Editor** operating mode
Remove the chips	Use this function to remove from the simulation those parts of the workpiece that were cut off during machining.	■ The **3D** model type

Workpiece before clean-up   Workpiece after clean-up
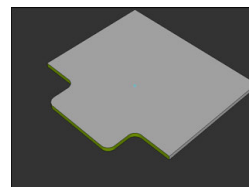
The Simulation settings window

Editor operating mode

The **Simulation settings** window contains the following areas in the **Editor** operating mode:

Area	Setting
General	■ **Model type** ■ **2.5D**: quick solid-model view ■ **None**: very fast simulation without solid-model view ■ **3D**: exact solid-model view ■ **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** (#50 / #4-03-1) ■ **Grinding** (#156 / #4-04-1) ■ **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 193 ■ **Prompt when opening a new simulation** If the toggle switch is active and you open the **Simulation** workspace in a new tab, the control shows the **Close current simulation** window. You can exit the active simulation or cancel the process. If the toggle switch is inactive, the control will not show the window. ⓘ If you open the **Simulation** workspace in 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	Setting
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 766 ■ **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 (#50 / #4-03-1) ■ Datum table ■ Preset table ■ Grinding tool table (#156 / #4-04-1) ■ Dressing tool table (#156 / #4-04-1) **Further information:** User's Manual for Setup and Program Run With the **Reset** button, the control selects the same tables for the simulation that are active for program run.

Program Run and Manual operating modes

The **Simulation settings** window contains the following setting in the **Program Run** and **Manual** operating modes:

Area	Setting
General	**Model type** ■ **2.5D**: quick solid-model view ■ **None**: very fast simulation without solid-model view

You can open the **Simulation settings** window only if there is no active traversing command. In this case, the **Control-in-operation** icon is shown in white.

Action bar
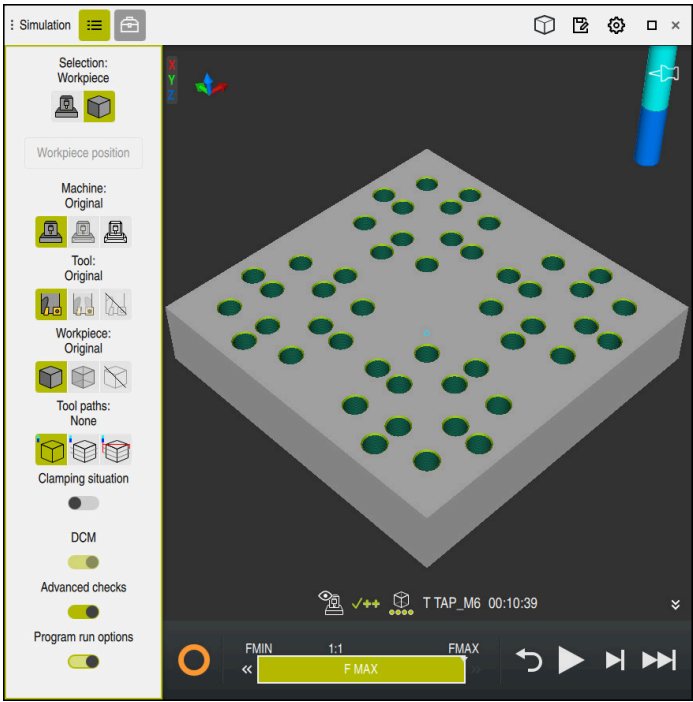
The **Simulation** workspace in the **Editor** operating mode

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 783
↩	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 784

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 408

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

The control visualizes the following entries of the turning-tool table (#50 / #4-03-1) in the simulation:

- **ZL**
- **XL**
- **YL**
- **RS**
- **T-ANGLE**
- **P-ANGLE**
- **CUTLENGTH**
- **CUTWIDTH**
- **KINEMATIC**

If the **ZL** and **XL** columns are defined in the turning tool table, the indexable insert is displayed and the base body is shown schematically.

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

The control visualizes the following entries of the grinding-tool table (#156 / #4-04-1) in the simulation:

- **R-OVR**
- **LO**
- **L-OVR**
- **LI**
- **B**
- **G**
- **R_SHAFT**
- **RV**
- **RV1**
- **RV2**
- **ALPHA**
- **GAMMA**
- **KINEMATIC**

  When you are dressing a tool, the control hides the tool carrier in the **Simulation** workspace.

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

The tool representation depends on the quality selected in the **Simulation settings** window. The control visualizes round tools as polygons. The lower the quality the fewer corners the polygon has, regardless of the tool diameter. Large tools also produce a more significant chord error, which means that the representation deviates more strongly from a circle.

Notes

> ### *NOTICE*
>
> **Danger of collision!**
>
> Even during simulation of an NC program, the control executes all SQL commands. Here, SQL commands may overwrite table values, for example, that also take effect in the **Program Run** operating mode. The overwritten values can lead to unexpected behavior or incorrect positioning during subsequent machining operations. There is a danger of collision.
>
> ▶ Skip SQL commands during simulations by using conditional jumps, for example
>
> ▶ Use **FN18: SYSREAD ID992 NR16** to check whether the NC program is active in a different operating mode or in **Simulation**

If the control is unable to machine the entire contour in turning cycles (#50 / #4-03-1), it will display locations with residual material in the simulation. The control displays the tool path in yellow instead of white and crosshatches the residual material.

The control will always display yellow tool paths and the crosshatching, independent of the selected mode, model quality, and display mode of the tool paths.

23.2 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
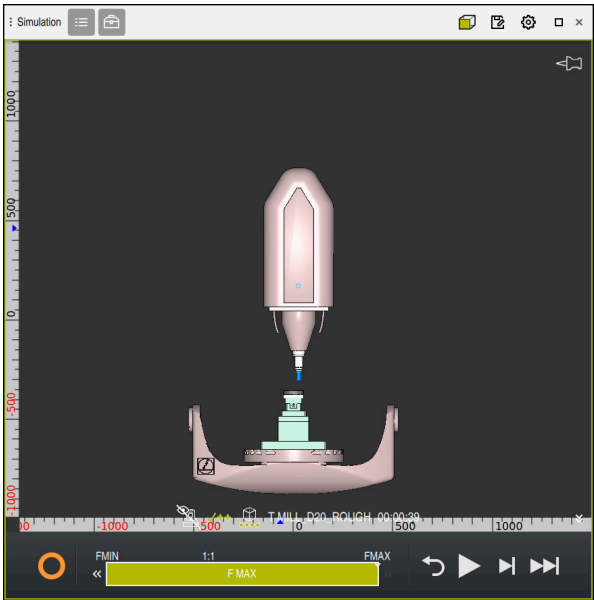
Front view of the simulated workpiece in the **Machine** mode

23.3   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 193

- Customizing STL files in **CAD Viewer** (#152 / #1-04-1)

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

Description of function
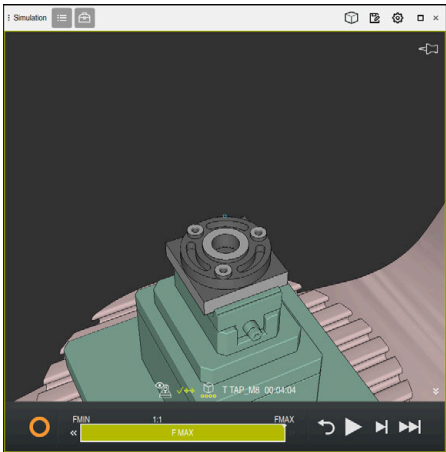
Simulated workpiece

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 768

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
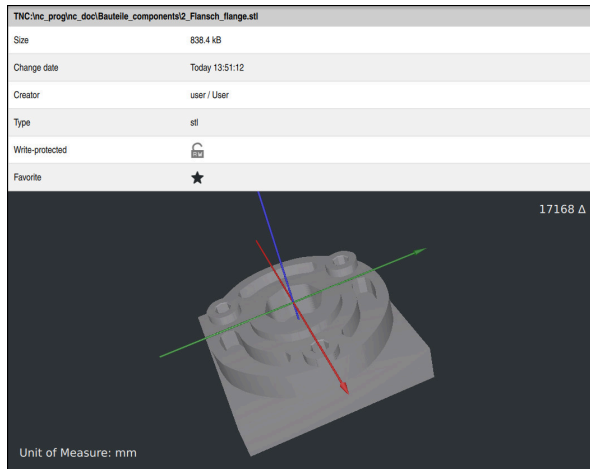
Simulated workpiece as saved STL file

23.3.1 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 768

23.4 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.
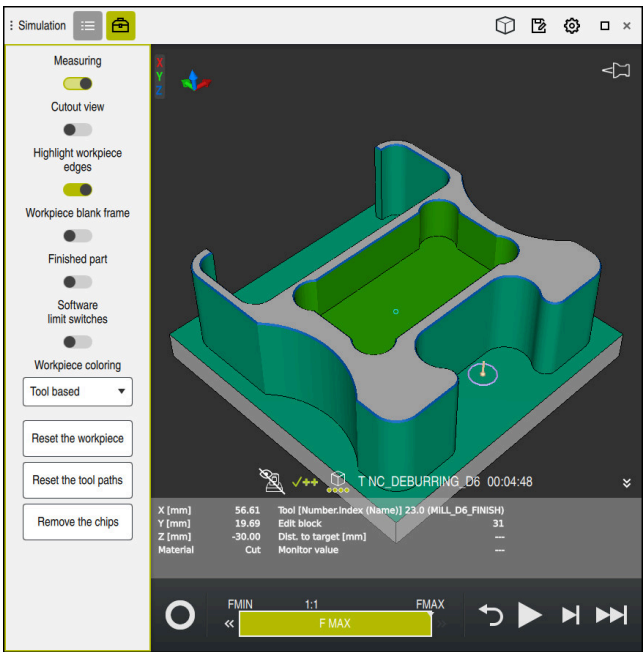
Measured point on simulated workpiece

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 316
■ 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

23.4.1 Measuring the difference between the workpiece blank and the finished part

To measure the difference between the workpiece blank and the finished part:

▶ Select an operating mode (e.g., **Editor**)

▶ Open an NC program with a workpiece blank and finished part defined in **BLK FORM FILE**

▶ Open the **Simulation** workspace

🧰   ▶ Select the **Tool options** column

          ▶ Activate the **Measuring** toggle switch

          ▶ Select the **Workpiece coloring** selection menu

[Model compari... ▼]   ▶ 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.

> **ℹ** 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

■ Depending on the selected quality in the **Simulation settings** window, the measurement result possibly does not match the real workpiece. The lower the quality the more strongly the simulated tool deviates from a circle, for example. During the simulation, this deviation (i.e.,the chord error), is transferred to the workpiece. Set the highest quality for simulation.

Further information: "Simulation of tools", Page 771

■ 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.

23.5 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.

23.5.1 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**.

▶ The selection menu allows you to select the area which the control shows after the cut such as **+X**

▶ Use the slider to specify the desired percent value

> The control simulates the workpiece with the selected sectional settings.
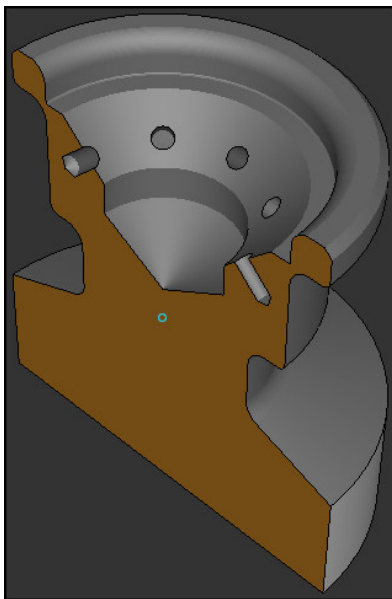
Simulated workpiece in the **Cutout view**

23.6 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 193

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
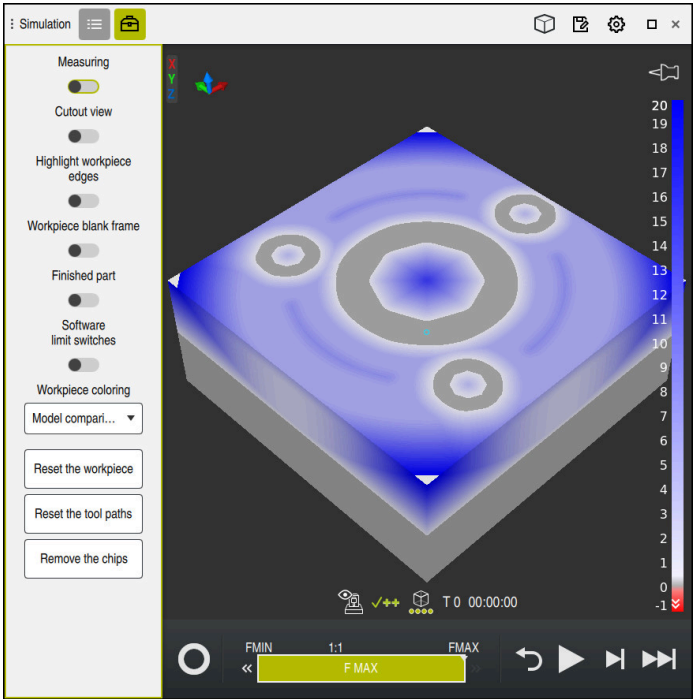
The control uses the **Model comparison** function to show the difference in material between the models being compared. The control uses a color transition from white to blue to show the difference in material. The more material there is covering the finished part model, the deeper the blue is. When 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 777

■ The result depends on the quality of the STL files and the quality selected in the **Simulation settings** window. The lower the quality the more strongly the simulation deviates from the real result. Set the highest quality for simulation.

23.7 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.

23.7.1 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".

23.8 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	Meaning
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**)

23.9 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 770
- Simulation speed
  Further information: "Simulation speed", Page 783

Description of function

This function can be used only in the **Editor** operating mode.
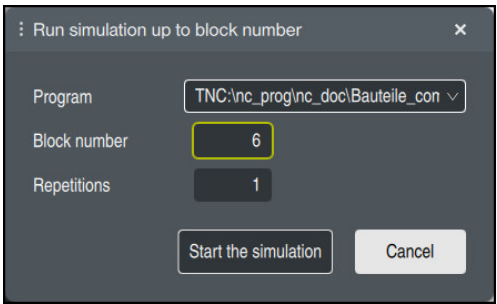
The **Run simulation up to block number** window with a defined NC block

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 285

23.9.1   Simulating an NC program up to a certain NC block

To simulate up to a specific NC block:

▶ Open the **Simulation** workspace

▶▶▶| ▶ Select **Run simulation up to block number**

> The control opens the **Run simulation up to block number** window.

▶ Use the selection menu in the **Program** field to specify the main program or called program

▶ Enter the number of the desired NC block in the **Block number** field

▶ If the block involves a program-section repeat, enter the number of the iteration of the program-section repeat in the **Repetitions** field

Start the simulation   ▶ Select **Start the simulation**

> The control simulates the workpiece up to the selected NC block.

Note

Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

24

Pallet machining and job lists

24.1 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 799

24.1.1 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 838

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 790

24.2 The Job list workspace

24.2.1 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 838
- The **Form** workspace for pallets
  Further information: "The Form workspace for pallets", Page 797
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 799

Requirement

- Batch Process Mngr. (#154 / #2-05-1) software option
  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 790

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 791

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 790
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 788 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 790
Sts	Machining status **Further information:** "Machining status", Page 791

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 processed in the **Program Run** 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
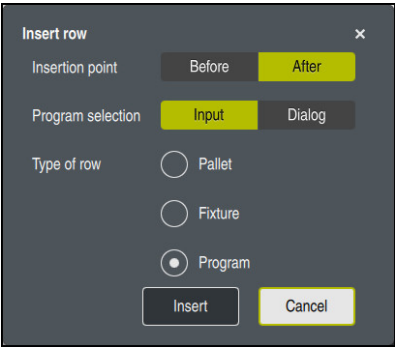
The **Insert row** window with the **Program** selection

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 797

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 792

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

- 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.

24.2.2 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 789
- Editing a pallet table in the **Form** workspace

  Further information: "The Form workspace for pallets", Page 797
- Contents of the pallet table

  Further information: "Pallet table *.p", Page 838

Requirements

- Batch Process Mngr. (#154 / #2-05-1) software option

  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
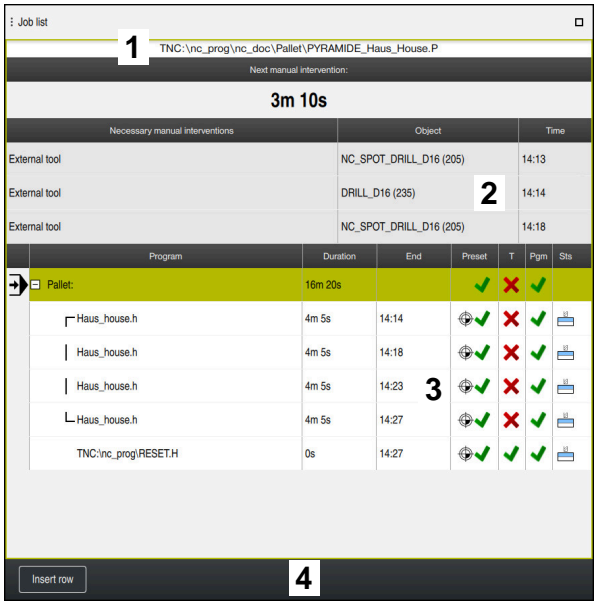
The **Job list** workspace with **Batch Process Manager** (#154 / #2-05-1)

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 795

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 790
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 788
	Machining method:
	■ Workpiece-oriented machining
	■ Tool-oriented machining
	Further information: "Machining method", Page 790
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 796
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 796
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 796
Sts	Machining status
	Further information: "Machining status", Page 791

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 ✔.

24.3 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 789
- Contents of the pallet table
  Further information: "Pallet table *.p", Page 838
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 799

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 the **Editor** or **Program Run** operating mode.

24.4 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 838
- 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 393
  - **M144** (#9 / #4-01-1)

    Further information: "Taking the tool offset into account in calculations with M144 (#9 / #4-01-1)", Page 586
  - **M101**

    Further information: "Automatically inserting a replacement tool with M101", Page 591
  - **M118**

    Further information: "Activating handwheel superimpositioning with M118", Page 570
  - Changing the pallet preset

    Further information: "Pallet preset table", Page 804

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 838

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 747

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 838

> ⚙ 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 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 working plane** or **M138**)

▶ Carefully test the NC program or program section in the **Single Block** 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**
- Cycle **800** (#50 / #4-03-1)
- 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 working plane** or **M138**)

▶ Carefully test the NC program or program section in the **Single Block** mode

---

- If you want to start machining again, change the W-STATUS to BLANK or remove the previous input.
- You can also monitor tool-oriented machining operations with the Process Monitoring software option. To do so you must have already used tool-oriented monitoring on the NC program.
  Further information: "Process monitoring (#168 / #5-01-1)", Page 501

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.

24.5   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 838
- 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

25

Tables

25.1 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 452

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "The Open File workspace", Page 452

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**, **Form and Document** workspaces in every application.

Further information: "The Table workspace", Page 813

Further information: "The Form workspace for tables", Page 820

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 747

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.
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.
Reset row	The control resets all data contained in the 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.

25.1.1 Editing the contents of tables

To edit the contents of a table:

▶ Select the desired table cell

Edit	▶ Enable **Edit**
	> The control enables the values for editing.

> ℹ️ To edit a table content, 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 cancel the process.

> ℹ️ If the **Edit** toggle switch is enabled, you can edit the contents both in the **Table** workspace and in the **Form** workspace.

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control enables you to transfer tables from previous controls to the TNC7 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: "Adapting files", Page 456

> ℹ️ Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., format errors).

Notes about machine parameters

Refer to your machine manual.

- Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. The machine parameter allows you to define columns as mandatory fields or reset them automatically to a default value. If this rule is not fulfilled, the control will display an information symbol.
- The machine manufacturer uses the machine parameter **CfgTableCellLock** (no. 135600) to define in which cases individual table cells will be blocked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.

25.2 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 452
- Available file types for tables

    Further information: "File types", Page 448

Description of function
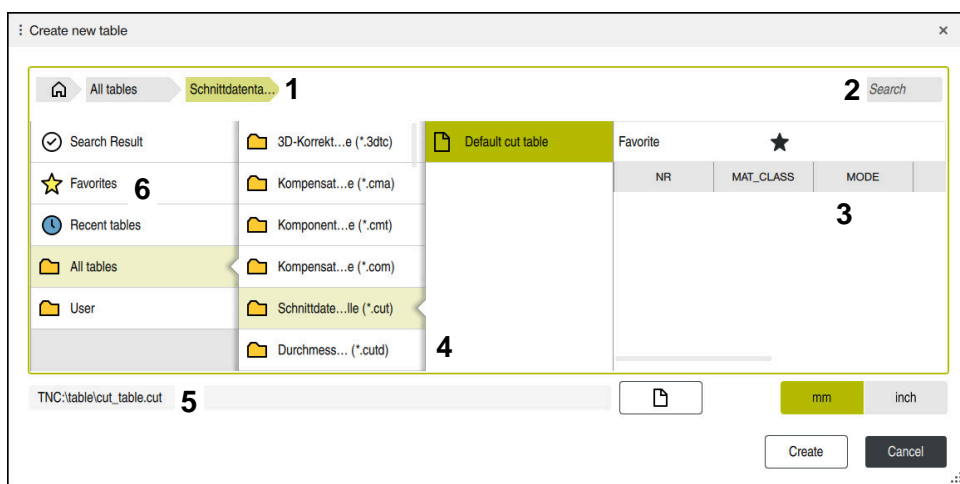
The **Create new table** window

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. You can edit the path or open a previous path from the History.

2   Content columns

    The control shows a folder and the available prototypes for each table type.

3   Searching

    You can search for any strings. The control displays the results under **Search Result**.

4   The control shows the following information and functions:

    - Add or remove a favorite
    - Preview

5   Unit of measure (mm or inches)

6   Path of the table to be created

7   Navigation column

The navigation column offers the following possibilities for navigation:

  ◼ **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**).

25.3 The Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. You can search in all tables and filter the table content.

Description of function
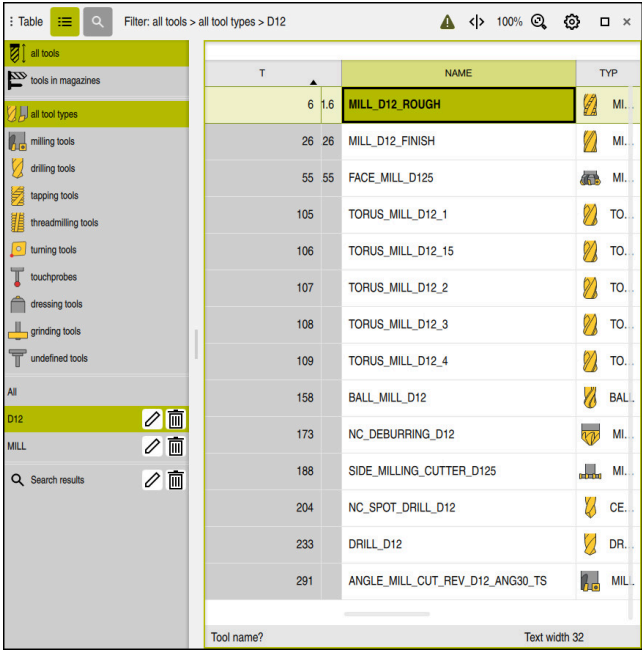
The **Table** workspace

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 in ascending or descending order.

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:** " Filter column in the Table workspace", Page 815
🔍 **CTRL + F**	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 818
⚠	■ Activate or deactivate the **"Rules not met" filter** The control displays only the rows that do not meet the rules defined by the machine manufacturer in **CfgTable-CellCheck** (no. 141300). ■ Open the **Data record consistency violations** window In these rows, the control shows the icon at the start of the row, even if the filter is inactive. The control displays a window showing how the row violates the rules.
🖊	**Edit table characteristics** **Further information:** "Modifying the properties of freely definable tables", Page 829
100%	Current size of the content Open or close the **Scale** 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 818
✏	Open the **Search** column and edit the selected filter Only in the **Filter** column **Further information:** "The Search column in the Table workspace", Page 818
🗑	Delete the selected filter Only in the **Filter** column **Further information:** " Filter column in the Table workspace", Page 815
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

Filter column in the Table workspace

The control offers default filters for the following tables:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

For all other tables, you can create user-defined filters.

Further information: "User-defined filters", Page 816

The **Filter** column in the **Table** workspace is divided into several filter groups. The control separates the filter groups with a white double line.

The control provides the following filter groups:

1 Default filters

Filters that are available by default for the respective table

In the **Tool management** application, the default filters comprise two filter groups

2 User-defined filters

Search processes saved as filters

Further information: "User-defined filters", Page 816

3 **Search results**

Results of the **Search** column

Further information: "The Search column in the Table workspace", Page 818

When you tap or click a filter once, the control activates only the selected filter in the corresponding area.

When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.

Further information: "Connecting conditions and filters", Page 817

The control highlights active filters green.

> ⚙   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.

Filters in the Tool management

The control provides the following default filters in the **Tool management**:

- **All tools**
- **Magazine tools**
- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Lathe tools** (#50 / #4-03-1)
- **Touch probes**
- **Dressing tools** (#156 / #4-04-1)
- **Grinding tools** (#156 / #4-04-1)
- **Undefined tools**

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- **All magazines**
- **spindle**
- **main magazine**
- **all pockets**
- **empty pockets**
- **occupied pockets**
- **Locked pockets**

Filters in the Presets table

The control provides the following default filters in the **Presets** table:

- **Show all**
- **Basic transformations**
- **Offsets**

User-defined filters

You can additionally create user-defined filters by saving a search.

Further information: "The Search column in the Table workspace", Page 818

The control only shows this filter group when you create a user-defined filter. In addition to the user-defined filter, the control provides the **All** filter possibility.

Further information: " Filter column in the Table workspace", Page 815

Connecting conditions 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 filter groups

  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 filter groups

  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

The control offers a search function in all tables.

You can define multiple search conditions in the **Search** function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
  Use the **Search in** selection menu to select the column.
- Operator if applicable (e.g., **Contains** or **Equal to (=)**)
  Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

> **i** If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use **Add** to add several conditions. The conditions will have a combined effect when you perform the search.
	You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a user-defined filter. You can assign any name to the filter. **Further information:** "User-defined filters", Page 816

An unsaved search works like a user-defined filter. If an unsaved search is active, the control highlights the **Search results** filter group green in the **Filter** column.
Further information: " Filter column in the Table workspace", Page 815

> **⚙** 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** 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
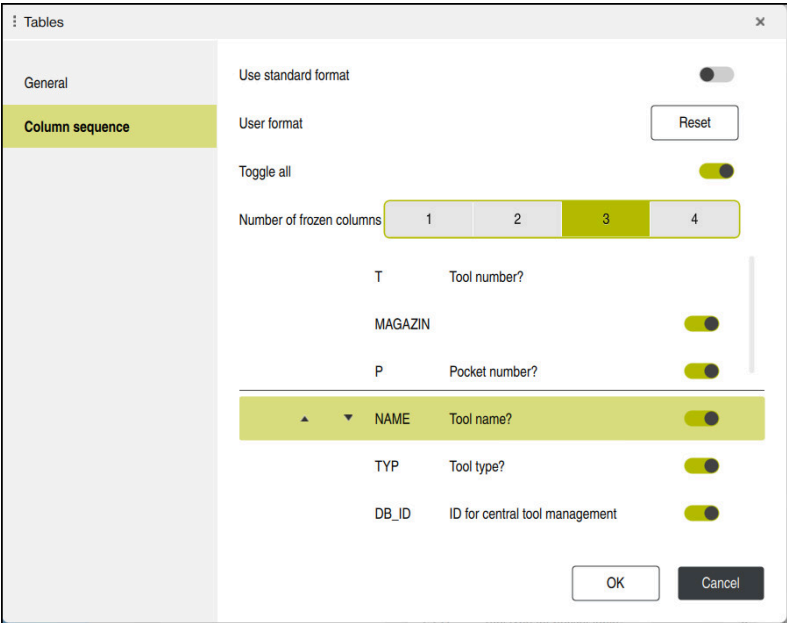
The **Tables** window

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.

25.4 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
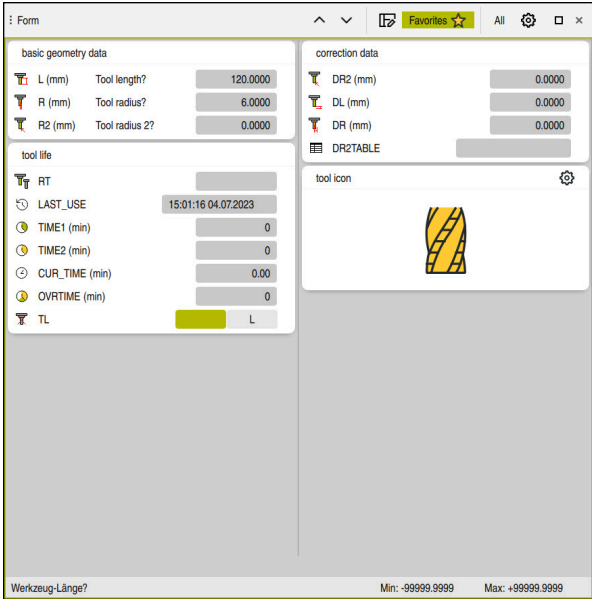
The **Form** workspace in the **Favorites** view

The control displays the following information for each parameter:

- Icon of the parameter, if applicable
- Name of the parameter
- Unit of measure as needed
- Parameter description
- Current value

The control displays the contents of specific tables in groups within the **Form** workspace.

> Refer to your machine manual.
> If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Buttons and icons

The **Form** workspace contains the following buttons, icons or shortcuts:

Buttons, icons or shortcuts	Meaning
∧ ∨ SHIFT + UP, SHIFT + DOWN	**Navigate** Navigate between table rows
🖉	**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
Favorites	In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites.
All	In this view the control shows all areas.
⚙	**Settings** ■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 822 ■ 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 822 ■ 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.
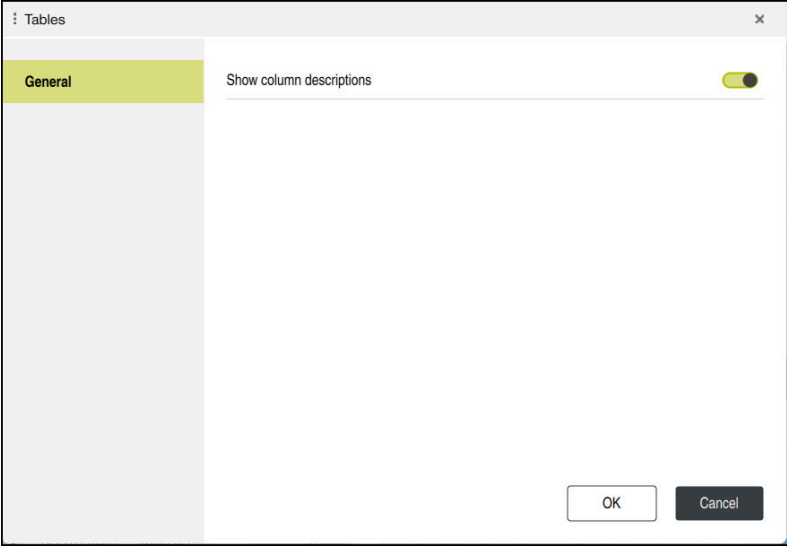
25.4.1 Adding a column in the workspace

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.
- For turning tools the icons also take into account the tool orientation and show where the relevant tool data will be in effect (#50 / #4-03-1).
  Further information: User's Manual for Setup and Program Run
- The control displays help graphics on how the parameters for grinding tools will be in effect (#156 / #4-04-1).
  Further information: "Grinding operations (#156 / #4-04-1)", Page 171

25.5 Accessing table values

25.5.1 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 824
TABDATA WRITE	Write a value to a table cell	Page 824
TABDATA ADD	Add a value to a table value	Page 826

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 599

- Tool table

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

- Compensation tables

  Further information: "Compensation tables", Page 842

- Reading values from freely definable tables

  Further information: "Reading a freely definable table with FN 28: TABREAD", Page 635

- Writing values to freely definably tables

  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 633

25.5.2 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 variable.

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 numerical or string parameters 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**

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ Functions ▶ TABDATATable access TABDATA ▶ TABDATA READ

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
READ	Read a table value
Parameter	Variable in which the control stores 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 Text or string parameter
KEY	Row number Number, text, or variable

25.5.3 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**, **QS** or named parameters as transfer parameters. 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 ▶ Table access TABDATA ▶ TABDATA WRITE

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
WRITE	Write 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
	Text or string parameter
KEY	Row number
	Number, text, or variable
= or **SET UNDEFINED**	Write the table value or assign the status **undefined**
Number, **Name** or **Parameter**	Table value
	Number, text, or variable
	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

---

25.5.4 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

You can define the value directly within the **TABDATA ADD** NC function or use numerical parameters as transfer parameters.

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 420

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 ▶ Table access TABDATA ▶ TABDATA ADD

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
ADD	Add 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
	Text or string parameter
KEY	Row number
	Number, text, or variable
Number	Value to be added
	Number or numerical parameter

25.6 Freely definable tables *.tab

Application

Freely definable tables can be read to from an NC program, and the program can also write any information to these tables. The NC functions **FN 26** to **FN 28** are available for this purpose.

Related topics

■ Variable functions **FN 26** to **FN 28**

Further information: "NC functions for freely definable tables", Page 633

Description of function
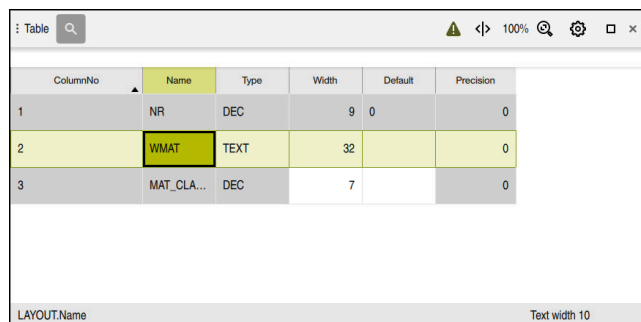
When you create a freely definable table, the control will provide various table templates for selection.

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 829
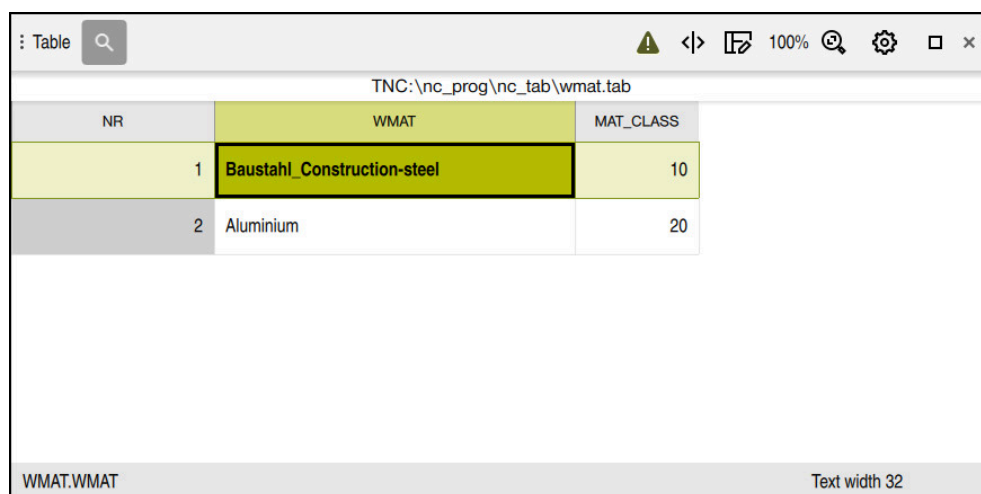
In the **LAYOUT** application, the control shows the columns of the table row by row.

Freely definable table in the **LAYOUT** application

Freely definable table in the **Table** workspace

Properties of a table column

When you change any table properties, each column has the following properties:

Column	Meaning
Name	Name of the column
Width	Maximum number of characters in the column
Default	Default value of each new row Optional input
Type	The control offers the following possible selections in the **Type** column: ■ **TEXT**: Text entry ■ **SIGN**: Algebraic sign + or − ■ **BIN**: Binary number ■ **DEC**: Positive integer ■ **HEX**: Hexadecimal number ■ **INT**: Integer ■ **LENGTH**: Floating-point number (mm or inch) ⓘ If you write values from an inch program to a freely definable table, the control converts the values. ⓘ If the unit of measure is inches, then the column has one more decimal place than you define. ■ **FEED**: Feed rate (mm/min or 0.1 ipm) ■ **IFEED**: Feed rate (mm/min or ipm) ⓘ If the unit of measure is inches, then the column has one more decimal place than you define. ■ **FLOAT**: Floating-point number ■ **BOOL**: Logical value ■ **INDEX**: Index ■ **TSTAMP**: Time and date with the format HH:MM:SS DD.MM.YYYY ■ **UPTEXT**: Text entry in capital letters ■ **PATHNAME**: Path name ⓘ In the columns with the data types **BIN**, **DEC** or **HEX** you can enter the values as binary numbers, positive integers or hexadecimal numbers. The control converts the entered values into the column's respective data type.
Precision	Maximum number of decimal places

25.6.1 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 **Edit**

▶ Select **Insert rows**

> The control opens the **Insert rows** window.

▶ Enter the **Column name**

▶ Select **Column type**

> The control displays a selection menu.

> **ℹ** You cannot change the column name or column type later.

▶ Select the desired column type

Further information: "Properties of a table column", Page 828

▶ 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.
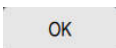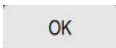
Notes

■ 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 655

■ 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 818

25.7   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**

25.7.1 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

    Edit   ▶ Activate **Edit**

                      ▶ Enter **Y**
                      > The control hides the point at the cycle call.

If you enter **Y** in the **FADE** column, you can use the **Skip block** toggle switch to skip this point in **Program Run** operating mode.
Further information: User's Manual for Setup and Program Run

25.8 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 832
- 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 columns **X**, **Y** and **Z** have the effect of a shift in the workpiece coordinate system **W-CS**. The values of columns **A**, **B**, **C**, **U**, **V** and **W** have the effect of offsets 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 316
	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 316
	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 316
	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 312
	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 312
	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 312
	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 312
	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 312
	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 312
	Input: **-99999.99999...+99999.99999**
DOC	**Comment on shift?**
	Entry: **Text width 16**

25.8.1 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:

Edit ●

▶ Activate **Edit**

▶ Select the value

▶ Edit the value

▶ Save the edited value, for example by selecting a different row

NOTICE
Danger of collision!
The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.
▶ Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
▶ Activate the datum or compensation value in the NC program again
▶ Carefully prove-out the NC program after changing the table values

25.9 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 835
- Table for tool materials **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 835
- Cutting data table ***.cut**

  Further information: "Cutting data table *.cut", Page 836
- Diameter-dependent cutting data table ***.cutd**

  Further information: "Diameter-dependent cutting data table *.cutd", Page 837

Related topics

- Cutting data calculator

  Further information: "Cutting data calculator", Page 756
- 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: **0...9999999**

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 835 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 835 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 210 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 211 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 835 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 835 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 210 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 211 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.

25.10 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 789
- Tool-oriented machining

  Further information: "Tool-oriented machining", Page 799

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...9999999**
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 799
	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 799 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 799 Input: **Text width 8**
SP-X	**Clearance height?** Clearance height in the X axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 799 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 799 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 799 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 799 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 799 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 799 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 799 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 799 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 799 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 788 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 788 Input: **0...99999**

25.11 Compensation tables

25.11.1 Overview

The control provides the following compensation tables:

Table	Further information
Compensation table ***.tco** Compensation in the tool coordinate system **T-CS**	Page 842
Compensation table ***.wco** Compensation in the working plane coordinate system **WPL-CS**	Page 844

25.11.2 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 418
- Contents of the compensation table ***.wco**
  Further information: "Compensation table *.wco", Page 844
- 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 323

Description of function

Any compensation in the compensation tables with the ***.tco** file name extension applies to the active tool. The table applies to all tool types. Therefore, columns that you may not need for your specific tool type will be displayed during creation.

Enter only those values that are relevant to your tool. If you compensate for values that are not present with the existing tool, the control issues an error message.

The compensation table ***.tco** provides the following parameters:

Parameter	Meaning
NO	Row number in the table Input: **0...999999999**
DOC	Comment Input: **Text width 16**
DL	**Tool length oversize?** Delta value for parameter **L** of the tool table Input: **-999.9999...+999.9999**
DR	**Tool radius oversize?** Delta value for parameter **R** of the tool table Input: **-999.9999...+999.9999**
DR2	**Tool radius oversize 2?** Delta value for parameter **R2** of the tool table Input: **-999.9999...+999.9999**
DXL	**Oversize in tool length 2?** Delta value for parameter **DXL** of the turning tool table Input: **-999.9999...+999.9999**
DYL	**Tool length oversize 3?** Delta value for parameter **DYL** of the turning tool table Input: **-999.9999...+999.9999**
DZL	**Oversize in tool length 1?** Delta value for parameter **DZL** of the turning tool table Input: **-999.9999...+999.9999**
DL-OVR	**Compensation of the overhang** Delta value for parameter **L-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DR-OVR	**Compensation of the radius** Delta value for parameter **R-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DLO	**Compensation of the total length** Delta value for parameter **LO** of the grinding tool table Input: **-999.9999...+999.9999**
DLI	**Compensation of the length to the inner edge** Delta value for parameter **LI** of the grinding tool table Input: **-999.9999...+999.9999**

25.11.3 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**).

The ***.wco** compensation tables are used mainly for turning (#50 / #4-03-1).

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 418
- Contents of the compensation table ***.tco**
  Further information: "Compensation table *.tco", Page 842
- 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 318

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**

25.12 *.3DTC compensation table

Application

In a ***.3DTC** compensation table, the control saves the radius deviation of ball-nose cutters from the nominal value at a defined inclination angle. For workpiece touch probes, the control saves the deflection behavior of the touch probe at a defined probing angle.

The control takes into account the saved data during the execution of NC programs and during probing.

Related topics

- 3D radius compensation depending on the tool's contact angle

  Further information: "3D radius compensation depending on the tool contact angle (#92 / #2-02-1) ", Page 439

- 3D calibration of the touch probe

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

Requirements

- Adv. Function Set 2 (#9 / #4-01-1) software option
- 3D-ToolComp (#92 / #2-02-1) software option

Description of function

The ***.3DTC** compensation tables must be saved in the **TNC:\system\3D-ToolComp** folder. In the **DR2TABLE** tool management column, you can then assign the tables to a tool.

You create a separate table for each tool.

A compensation table provides the following parameters:

Parameter	Meaning
NR	Sequential row number in the compensation table
	The control evaluates a maximum of 100 rows in the compensation value table.
	Input: **0...9999999**
ANGLE	Inclination angle of tools or probing angle of workpiece touch probes
	Input: **-99999.999999...+99999.999999**
DR2	Radius deviation from the nominal value or deflection of the touch probe
	Input: **-99999.999999...+99999.999999**

26

Overviews

26.1 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

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

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