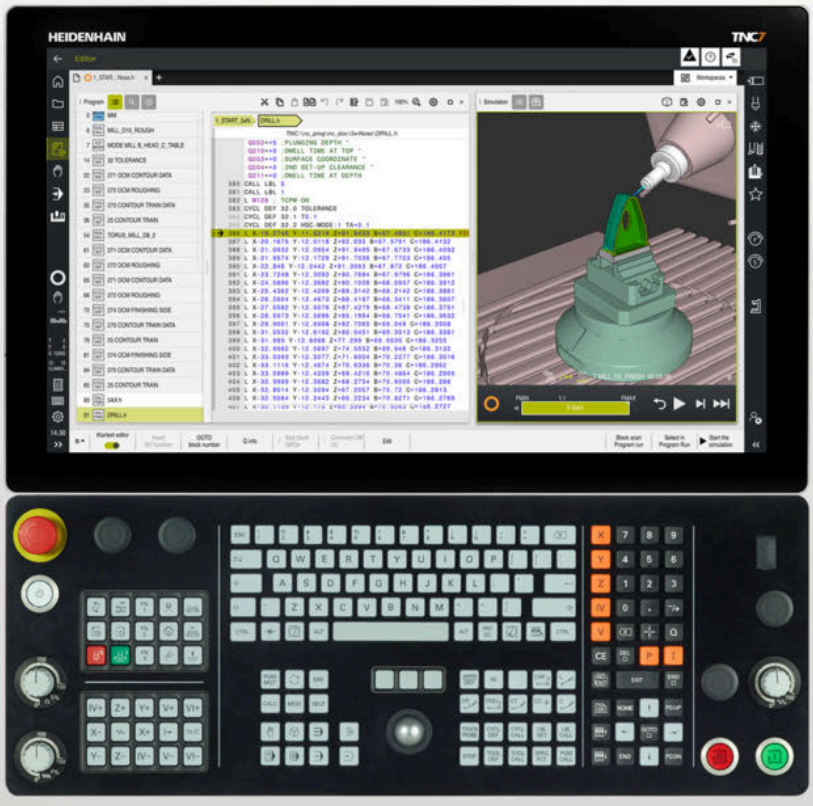
HEIDENHAIN

TNC7

User's Manual
Setup and Program Run

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.
	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 Status displays

Topic	Description
The **Status** workspace	In the **PROCMON** tab of the **Status** workspace, the control shows information on process monitoring (#168 / #5-01-1) in the **Program Run** operating mode. When process monitoring for the NC program is set up completely, you will get a compact overview of the current machining process. **Further information:** "The PROCMON (#168 / #5-01-1) tab", Page 139
	In the **PGM** tab of the **Status** workspace, you can open the **Counter settings** window in the **Parts counter** area. You can enter the current count and the target value for the counter. **Further information:** "The PGM tab", Page 135
	In the **QPARA** tab of the **Status** workspace, the **Variables** area has been added. In this area, the control shows the values of the named parameters you define. **Further information:** "The QPARA tab", Page 140

1.1.4 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.

1.1.5 Monitoring (#168 / #5-01-1)

Topic	Description
Process monitoring (#168 / #5-01-1)	The control provides the **Filter** column, in which you can filter the monitored operations by date or status, for example. **Further information:** "Areas of the Process Monitoring workspace", Page 307
	The control provides the **Form** column with settings or the monitoring tasks. The contents and options depend on the active table mode. **Further information:** "Settings in the Form column", Page 316
	The table in the **Process Monitoring** workspace has two modes: **Setup table** and **Runtime table**. You can switch between the modes with icons. In **Setup table** mode, you can see all monitoring sections of the NC program and the defined monitoring tasks. In **Runtime table** mode, you see the machining processes being monitored and the corresponding information.
	Process monitoring includes the monitoring task **Feed per tooth — Display**. **Further information:** "Overview of monitoring tasks", Page 326
	The **MinMaxTolerance**, **Standard deviation** and **Absolute deviation** processes have been combined in one **Tunnel** process. The previous processes continue to exist as parameterization options. **Further information:** "Tunnel", Page 331

1.1.6 Opening CAD files with CAD Viewer

Topic	Description
Rotate	The arrow icon in **CAD Viewer** now includes the **Rotate** mode. The **Rotate** mode is active by default and enables pure touch operation. **Further information:** "Screen layout", Page 339
Output options	The **Setting for whether comments are written to NC output files.** icon has been added to **CAD Viewer**. Via this icon you can select whether **CAD Viewer** transfers information on the workpiece blank, the datum and the preset to the NC program

1.1.7 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.
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.
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).
Showing NC blocks	You can show hidden NC blocks with the **BACKSPACE** key.
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**
	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.

1.1.8 Touch probes

Topic	Description
Set Up Touch Probes	The **Overview of touch probes** menu item of the **Settings** application replaces the **Set Up Touch Probes** HEROS function. **Further information:** "Overview of touch probes menu item", Page 569
	You can set up touch probes using **TNCdiag**. **Further information:** "TNCdiag", Page 623

1.1.9 Touch probe functions in the Manual operating mode

Topic	Description
Set up the workpiece (#159 / #1-07-1)	You can choose an NC program within the **Set up the workpiece** function. The control generates the 3D model from the workpiece blank definition of the NC program. **Further information:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398
	If the active unit of measure is inch, the control converts mm to inches within the **Set up the workpiece** function.
	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. **Further information:** "Error estimate diagram", Page 403
The **Setup** application	The control provides the **Remove last measurement** button to repeat an erroneous touch point, for example.

1.1.10 Program run

Topic	Description
Automatic program start	Use the **Automatic program start** function to define the moment from which the control autonomously executes an NC program. **Further information:** "Automatic program start", Page 433

1.1.11 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 448
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 445

1.1.12 Electronic handwheel

Topic	Description
HR 180 panel-mounted handwheel in the operating panel	The control supports the **HR 180** panel-mounted handwheel. **Further information:** "Overview", Page 520
Wireless handwheel	You can set up wireless handwheels using **TNCdiag**. **Further information:** "TNCdiag", Page 623

1.1.13 Integrated functional safety (FS)

Topic	Description
F limited toggle switch	The **F limited** toggle switch has been removed.
Test status of the axes	You can reset the test status of individual axes or all axes in the **Axis status** menu item of the **Settings** application. To reset the test status of axes, you need the NC.ApproveFsAxis right. This right is only available if user administration is active. **Further information:** "The Axis status menu item", Page 553

1.1.14 The Settings application

Topic	Description
Menu item **Adjustment of analog voltage offset**	In the **Adjustment of analog voltage offset** menu item, the control shows all connected analog axes, whether the axes are in closed-loop control, and the current voltage offset. If the axis permits it, you can change the value of the voltage offset. **Further information:** "Adjustment of analog voltage offset menu item", Page 572
SIK menu item	In the **SIK Information** area, the control shows whether it features **SIK** or **SIK2**. **Further information:** "The SIK menu item", Page 565
DNC menu item	The **Fingerprint of the host key** function has been added. If you press the **Show** button, the control shows a unique ASCII graphic, comparable to a fingerprint. If you configure a secure connection, you can compare this ASCII graphic with a graphic included in the client application. That way you can ensure the connection is set up with the correct control. **Further information:** "The DNC menu item", Page 598
OPC UA NC Server (#56-61 / #3-02-1*)	On controls with **SIK2**, you can enable up to ten (instead of six) OPC UA connections. **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590
	The machine manufacturer can permit login with a user name and password, for example for client applications that do not support login with a user certificate. **Further information:** "Login options", Page 592
	In the **OPC UA** menu item of the **Settings** application, the control indicates the options available to the current user for logging in. **Further information:** "The OPC UA (#56-61 / #3-02-1*) menu item", Page 595
	Client applications can change the counter reading with the aid of the OPC UA and the NC.RemoteOperator role.

1.1.15 Machine parameters

Topic	Description
Configuration editor	In the **Configuration** window, the **Display in tree view** toggle switch has been added. With this toggle switch, you can display the machine parameters in a tree view instead of the structure view. **Further information:** "The Configuration window", Page 630
	The **Go to number** button has been added; you can use this button to enter the number of a machine parameter and move directly to this parameter. **Further information:** "Machine parameters", Page 626
	In the configuration editor, you can insert objects with the key combination **CTRL + N**.
	If the table view of the configuration editor is active, you can change the width of the active column via the **Change column width** icon.
SYM (**SEQ**) tilting solution	Via the optional machine parameter **prohibitSEQ** (no. 201209), the machine manufacturer defines whether only **SYM** or also **SEQ** are available for selection.
Reactions of component monitoring (#155 / #5-02-1)	Use the machine parameter **enforcedReactions** (no. 129403) to define the component monitoring reactions to be carried out by the control. By default, the control carries out all reactions. This machine parameter replaces the previous machine parameters **enforceReaction** (no. 129401) and **showWarning** (no. 129402).
Process monitoring (#168 / #5-01-1)	With the optional machine parameter **autoExportType** (no. 141602), you can define a file type that the control will automatically use to export recordings.
Conversational language	The control supports Japanese as a conversational language. The desired conversational language can be selected via machine parameters **ncLanguage** (no. 101301) and **plcDialogLanguage** (no. 101302).
	Use the optional machine parameter **noRebootDialog** (no. 101306) to define whether, after a change of the conversational language, the control displays a restart message.
Override controller	Via the optional machine parameter **ocColourScheme** (no. 103413), the machine manufacturer defines an alternative color scheme for the override controller.

1.1.16 User administration

Topic	Description
Rights	The HEROS.NormalUser role includes the HEROS.MountUSBDevices right. You need this right to be able to connect a USB device to the control. **Further information:** "User administration roles and rights", Page 687
	The control provides the NC.OpmodeSingleStep right, which allows you to execute NC programs in **Single Block** mode.
	The control provides the NC.OpcUaPwAuth right. With this right and a user name and password, you can log in to the **OPC UA NC Server** (#56-61 / #3-02-1*).
	The control provides the NC.OpcUaPwAuthOnlyMachineNet right. With this right and a user name and password, you can log in to the **OPC UA NC Server** (#56-61 / #3-02-1*) via the **eth1** network interface.
Machine parameters	The machine manufacturer defines which machine parameters are saved user-specifically by the control when user administration is active. These machine parameters can be changed at any time without, for example, having to restart the control.
Buttons	When user administration is active, the control provides the **Lock screen**, **Change user** and **Log off user** buttons in the **Start/Login** application. **Further information:** "Logging on with user administration", Page 654
Function users pre-defined by the machine manufacturer	HEIDENHAIN provides templates for the **oemreadonly** and **oemautomation** function users, which the machine manufacturer can activate. These function users can be used to set up and operate external systems (e.g., robots). Refer to your machine manual. **Further information:** "User administration", Page 635

1.1.17 HEROS operating system

Topic	Description
HEROS menu	The **Hostkey** HEROS tool has been added; the control can use this tool to display a unique ASCII graphic, similar to a fingerprint. If you configure a secure connection, you can compare this ASCII graphic with a graphic included in the client application. That way you can ensure the connection is set up with the correct control.

1.1.18 Accessories

Topic	Description
ITC	Using the Batch Process Manager BPM (#154 / #2-05-1), you can process pallet tables on the ITC.
Handwheel	The **HRA 180** handwheel adapter and the **HR 180** panel-mounted handwheel replace the **HRA 110** and **HR 150** products.

1.2 Modified or extended functions

1.2.1 Status displays

Topic	Description
The **Status** workspace	The **MON** tab of the **Status** workspace has been renamed as **COMPMON** (#155 / #5-02-1).
	The machine manufacturer defines the contents to be shown in the **COMPMON** tab of the **Status** workspace (#155 / #5-02-1). **Further information:** "The COMPMON (#155 / #5-02-1) tab", Page 131
	If the values of Cycle **32 TOLERANCE** are restricted by Dynamic Collision Monitoring DCM (#40 / #5-03-1), the control will show the text **DCM limited** in the **CYC** tab of the **Status** workspace after the value concerned. **Further information:** "CYC tab", Page 132
Display of the axis names	If the name of an axis has more than two characters, the control will adapt the width of the orange background.
Program runtime in the status overview on the TNC bar	If the TNC bar is minimized, the control indicates the program runtime with the units **m** and **s** or **h** and **m**.

1.2.2 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. **Further information:** "Error estimate diagram", Page 261
Combining fixtures	The individual components of a combined fixture retain assigned attributes (e.g., colors). **Further information:** "Combining fixtures in the New Fixture window", Page 271
Advanced checks	The **Workpiece collision** check has been renamed to **Collision between workpiece and tool**.

1.2.3 Monitoring (#168 / #5-01-1)

Topic	Description
Process monitoring (#168 / #5-01-1)	By default, the control displays six monitoring tasks. **Further information:** "Overview of monitoring tasks", Page 326
	Process monitoring shows suggestion type notes (e.g., **Consider deleting all records for NC program**). **Further information:** "Areas of the Process Monitoring workspace", Page 307
	The **Signal display** icon allows you to switch between the signal curve and the resulting value during program run.
	The selection options for saving the process data have been enhanced. If you select **Interval: Record each nth operation and critical operations**, the control will only record the process data of, for example, every fourth machining process and of processes affected by interruptions.
	If process monitoring is active and you interrupt the program run within a monitoring section, the control will deactivate process monitoring only for this monitoring section. For the following monitoring section, process monitoring is active again.

1.2.4 Touch probe functions in the Manual operating mode

Topic	Description
Working plane is inconsistent! (#8 / #1-01-1) window	The **3-D ROT Apply status** function no longer transfers only the current positions of the rotary axes to the **3-D rotation** window. For a consistent machining plane, the control takes over the status of the tilting function and the spatial angle from the **Program Run** operating mode or the **MDI** application, if necessary.
Set up the workpiece (#159 / #1-07-1)	By default, the control positions the 3D model to the active preset.
	If the active preset contains at least one spatial angle, the control by default selects the **6D** probing mode. **Further information:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398
Change the preset icon	The position of the **Change the preset** icon in the probing functions has been modified. When you select a touch probe function, the control immediately displays a message for checking the preset.

1.2.5 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.

1.2.6 Program run

Topic	Description
Block scan	If program run is canceled during a program section repeat or program loop, the control provides the number of the repeat as point of interruption. **Further information:** "Block scan for mid-program startup", Page 424
	If the machine parameter **operatingTimeReset** (no. 200801) is defined with the value **TRUE**, the effect has changed. The control no longer resets the program run time when you start block scan to the point of interruption after a program cancellation.
Open in the editor button	In **Editor** operating mode, the control selects the same NC block that is currently selected in **Program Run** operating mode.

1.2.7 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.
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.

1.2.8 Electronic handwheel

Topic	Description
Rotational speed in the display handwheel	The display of the handwheel always shows the spindle speed of the current channel (for example for a machine with several tool spindles). The handwheel also shows the speed of the rotary table (#50 / #4-03-1). **Further information:** "Handwheel with display", Page 525
Wireless handwheel	The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

1.2.9 Integrated functional safety (FS)

Topic	Description
Self-test of the control	If the self-test of the control is active, the control displays an icon in the information bar.
Internally monitored axes	Internally monitored axes can be activated and deactivated at run-time (e.g., interchangeable heads). The machine manufacturer must configure the activation and deactivation.
Check axis positions	In the **Referencing** workspace, you can switch as desired between the **Referencing** and **Check axis positions** modes.

1.2.10 The Settings application

Topic	Description
PKI Admin	The **PKI Admin** has been reorganized. The **Advanced settings** tab has been removed and the settings were transferred to the appropriate tabs. **Further information:** "PKI Admin", Page 588
The **VNC** menu item	If a VNC connection is active, the control shows the icon of the connection status in the information bar. **Further information:** "The VNC menu item", Page 605
TNCscope	The **TNCscope** application can only be opened with machine manufacturer rights.

1.2.11 Machine parameters

Topic	Description
Configuration editor	The key combination **CTRL + F** opens the **Search** column in the configuration editor.
	No longer needed machine parameters of the TNC 640 have been removed from the configuration editor of the TNC7.

1.2.12 User administration

Topic	Description
Connection to Windows domain	To join a Windows domain, the IT administrator must set up a function user. You can no longer join a Windows domain with a computer account.
	The control no longer automatically checks for every connection whether all of the required roles in the domain have been defined. Press the **Check missing role definitions** button to start the check. **Further information:** "Connection to Windows domain", Page 648
Rights	The NC.OpmodeProgramRun right now only comprises Program run in Full Sequence mode, no longer in **Single Block** mode.
Function users pre-defined by the machine manufacturer	The maximum number of function users has been increased from 16 to 32.

1.2.13 HEROS operating system

Topic	Description
Firewall	The firewall has been revised. You can protect every interface and source with the firewall.
HEROS menu	The **TNCscope** application can only be opened with machine manufacturer rights.

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)

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 60

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.

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 49

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 60

Related topics

- The **Help** workspace

  Further information: Programming and Testing User's Manual

Requirement

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

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

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

> You can also open the information page manually by selecting the **index.html** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired 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.

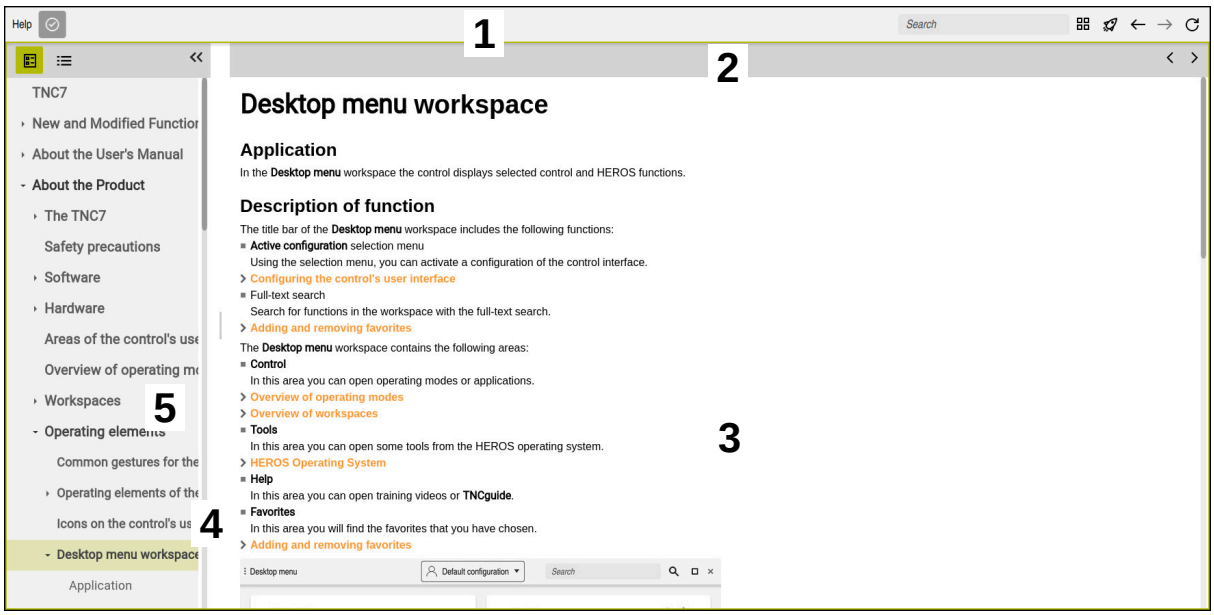
Further information: "The Help application", Page 54

Further information: Programming and Testing User's Manual

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

Further information: "Icons", Page 55

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 55

2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 55

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 56
	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 57

Context-sensitive help

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

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

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

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

Further information: "The Help application", Page 54

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

Further information: Programming and Testing User's Manual

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 52

- 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: Programming and Testing User's Manual

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: Programming and Testing User's Manual

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

Further information: "Hardware enhancements", Page 80

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.

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

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

Contribute actively to increasing the safety by reporting any missing or misleading information.

Further information: "Contacting the editorial staff", Page 58

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.

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

DANGER

Caution: hazard to the user!

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

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

DANGER

Caution: hazard to the user!

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

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

WARNING

Caution: hazard to the user!

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Caution: Danger to the tool and workpiece!

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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.

The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 67

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

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

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

NC software number	Product
817620-19	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: "Viewing of software options", Page 566

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.

> 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: "The SIK menu item", Page 565

Overview

> Keep in mind that particular software options also require hardware extensions.
>
> **Further information:** "Hardware", Page 76

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:** Programming and Testing User's Manual ■ 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:** Programming and Testing User's Manual ■ 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:** Programming and Testing User's Manual ■ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** Programming and Testing User's Manual ■ 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:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
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:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348
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:** "Global Program Settings GPS (#44 / #1-06-1)", Page 287
Adaptive Feed Contr. (#45 / #2-31-1)	**Adaptive Feed Control (AFC)** This software option enables an automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases and reduces the feed rate as the load increases. With AFC you can shorten machining times without adapting the NC program, while at the same time preventing machine damage from overload. **Further information:** "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276

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:** Programming and Testing User's Manual
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:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590
4 Additional Axes (#77 / #6-01-1*)	**Four additional control loops** **Further information:** "**Control Loop Qty.** (#0-7 / #6-01-1*)", Page 68
8 Additional Axes (#78 / #6-01-1*)	**Eight additional control loops** **Further information:** "**Control Loop Qty.** (#0-7 / #6-01-1*)", Page 68

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:** Programming and Testing User's Manual
Ext. Tool Management (#93 / #2-03-1)	**Extended tool management** This software option extends tool management by the two tables **Tooling list** and **T usage order**. The tables show the following contents: ■ The **Tooling list** shows the tool requirements of the NC program or pallet to be run **Further information:** "Tooling list (#93 / #2-03-1)", Page 502 ■ The **T usage order** shows the tool order of the NC program or pallet to be run **Further information:** "T usage order (#93 / #2-03-1)", Page 500 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 **Further information:** "Turning tool table toolturn.trn (#50 / #4-03-1)", Page 468 ■ **FUNCTION SHAPING** for contour planing ■ 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 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
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:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

Software option	Definition and application
Collision Monitoring (#140 / #5-03-2)	**Dynamic Collision Monitoring DCM version 2** This software option includes all functions of the Collision Monitoring software option (#40 / #5-03-1). In addition, this software option provides the following features: ■ Collision monitoring of fixtures **Further information:** "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256 ■ Define reduced minimum distance between fixture and tool
Cross Talk Comp. (#141 / #2-20-1)	**Compensation of axis couplings (CTC)** Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adapt. Contr. (#142 / #2-21-1)	**Position Adaptive Control (PAC)** Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adapt. Contr. (#143 / #2-22-1)	**Load Adaptive Control (LAC)** Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adapt. Contr. (#144 / #2-23-1)	**Motion Adaptive Control (MAC)** Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Contr. (#145 / #2-30-1)	**Active Chatter Control (ACC)** With this software option the chatter tendency of a machine used for heavy machining can be reduced. The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the metal-removal rate can be increased by more than 25%. **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 286
Machine Vibr. Contr. (#146 / #2-24-1)	**Vibration damping for machines (MVC)** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**
CAD Model Optimizer (#152 / #1-04-1)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356

Software option	Definition and application
Batch Process Mngr. (#154 / #2-05-1)	**Batch Process Manager (BPM)** This software option makes it easy to plan and execute multiple production jobs. By extending and combining the pallet management and extended tool management functions (#93 / #2-03-1), 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:** Programming and Testing User's Manual
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:** Programming and Testing User's Manual
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
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

Software option	Definition and application
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:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398
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:** "Process monitoring (#168 / #5-01-1)", Page 302

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: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590

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 66

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

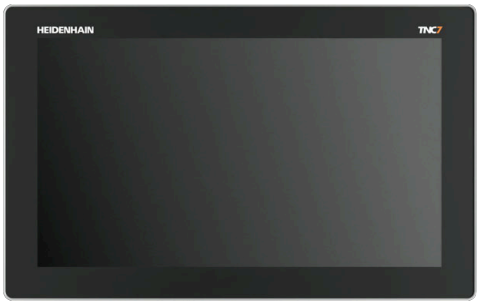
Further information: "Hardware enhancements", Page 80

Further information: "Software options", Page 67

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 89

Further information: "Operating elements of the keyboard unit", Page 89

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: "The Settings application", Page 557

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

The following cleaning agents are permitted for the screen:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

The following cleaning agents are prohibited for the screen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

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

Cleaning the keyboard unit

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: risk of property damage

Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.

- ▶ Use permitted cleaning agents only
- ▶ Use a clean, lint-free cleaning cloth to apply the cleaning agent

The following cleaners are permitted for the keyboard unit:

- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:

- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

Wear 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

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

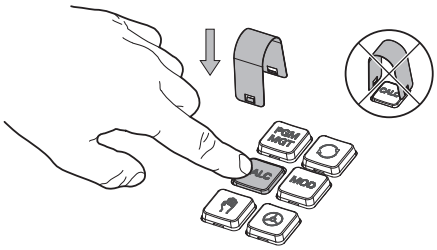
- ▶ Apply a small amount of the cleaning agent onto a cleaning cloth
- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

Exchanging keycaps

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

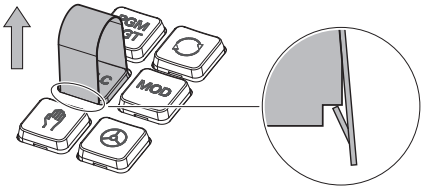
IP54 protection cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

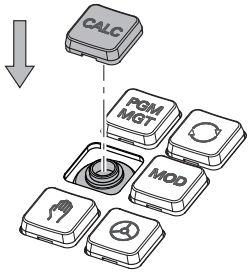


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

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



- Pull off the keycap



- Place the keycap onto the seal and push it down

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

- Verify proper seating and correct functionality

3.4.2 Hardware enhancements

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

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

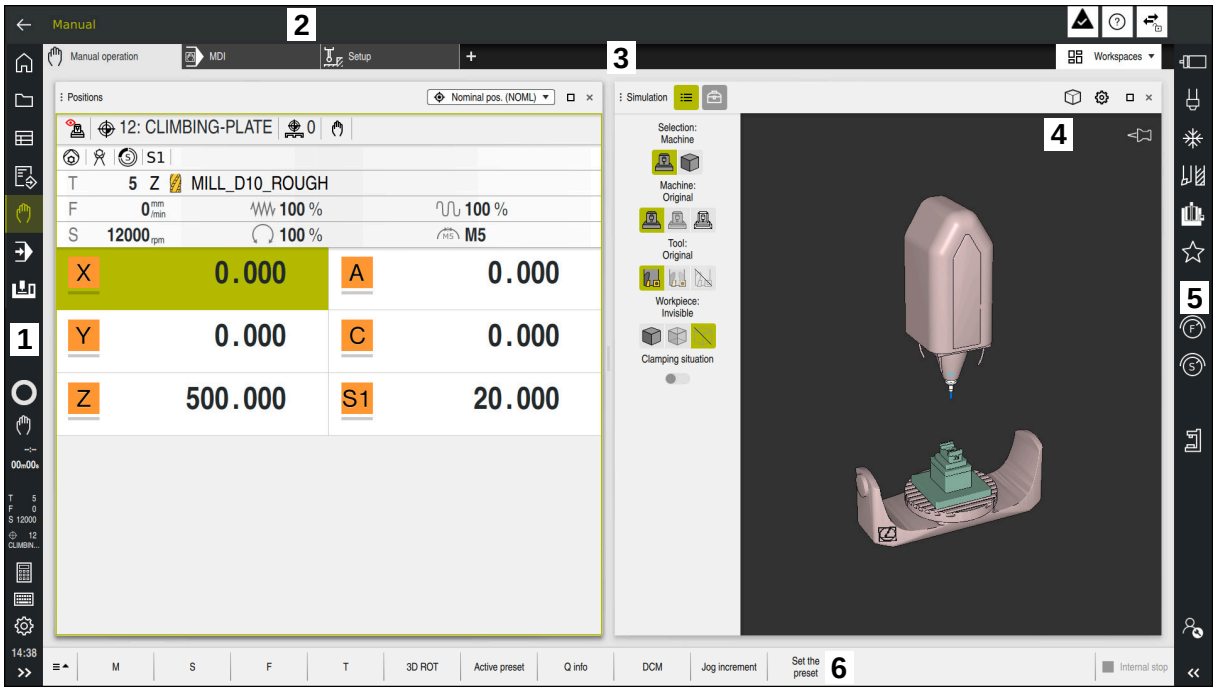
Keep in mind that particular hardware enhancements require additional software options.

Further information: "Software options", Page 67

Hardware enhancements	Definition and application
Electronic handwheels	You use this enhancement for exact manual positioning of machine axes. The wireless portable variants improve ergonomics and increase versatility. The handwheels have the following differing features: ■ Portable or installed in the machine operating panel ■ With or without display ■ With or without functional safety Electronic handwheels, for example, greatly simplify workpiece setup. **Further information:** "Electronic handwheel", Page 519
Workpiece touch probes	The control uses this enhancement for automatic and precise detection of workpiece positions and misalignments. The workpiece touch probes have the following differing features: ■ With radio or infrared transmission ■ With or without cable Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run. **Further information:** "Touch probe functions in the Manual operating mode", Page 373
Tool touch probes	With this extension, the control can measure tools automatically and precisely, directly in the machine. Tool touch probes have the following differing features: ■ Contact-free or tactile measurement ■ With radio or infrared transmission ■ With or without cable Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual

Hardware enhancements	Definition and application
Vision systems	Use this enhancement to inspect the tools used. With the VT 121 vision system, you can visually inspect the cutting edges during program run without removing the tool. The vision systems help to avoid damage during program run, thus preventing unnecessary costs. **VTC User's Manual** All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID: 1322445-xx
Additional operating stations	This enhancement adds a second screen, to facilitate operation of the control. The additional ITC (industrial thin client) operating stations are differentiated by their intended use: ■ The ITC 755 is a compact, additional operating station that mirrors the control's main screen, making it possible to operate the control. ■ The ITC 860 is an auxiliary screen that increases the area of the main screen. This allows multiple applications to be viewed simultaneously. By adding a keyboard unit, the ITC 860 can be used as a full-fledged additional operating station. The additional operating stations increase operator comfort, especially on large machining centers.
Industrial PC	You use this enhancement to install and run Windows-based applications. With Remote Desktop Manager (#133 / #3-01-1), you can display applications on the control screen. **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608 The industrial PC is a secure and powerful alternative to external PCs.
Override controller	This extension allows you to define breakpoints at which the control stops during program run (e.g., before a tilting function). The override controller enables the feed rate or rapid traverse value to be changed as well as starting or continuing the NC program. **Further information:** "Override controller", Page 537

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 84
- Status overview

  Further information: "Status overview on the TNC bar", Page 125
- Calculator

  Further information: Programming and Testing User's Manual
- Screen keyboard

  Further information: "Virtual keyboard of the control bar", Page 362
- 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

  Further information: "Message menu on the information bar", Page 365
- Symbols

  Further information: "Icons on the control's user interface", Page 97

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 85

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.

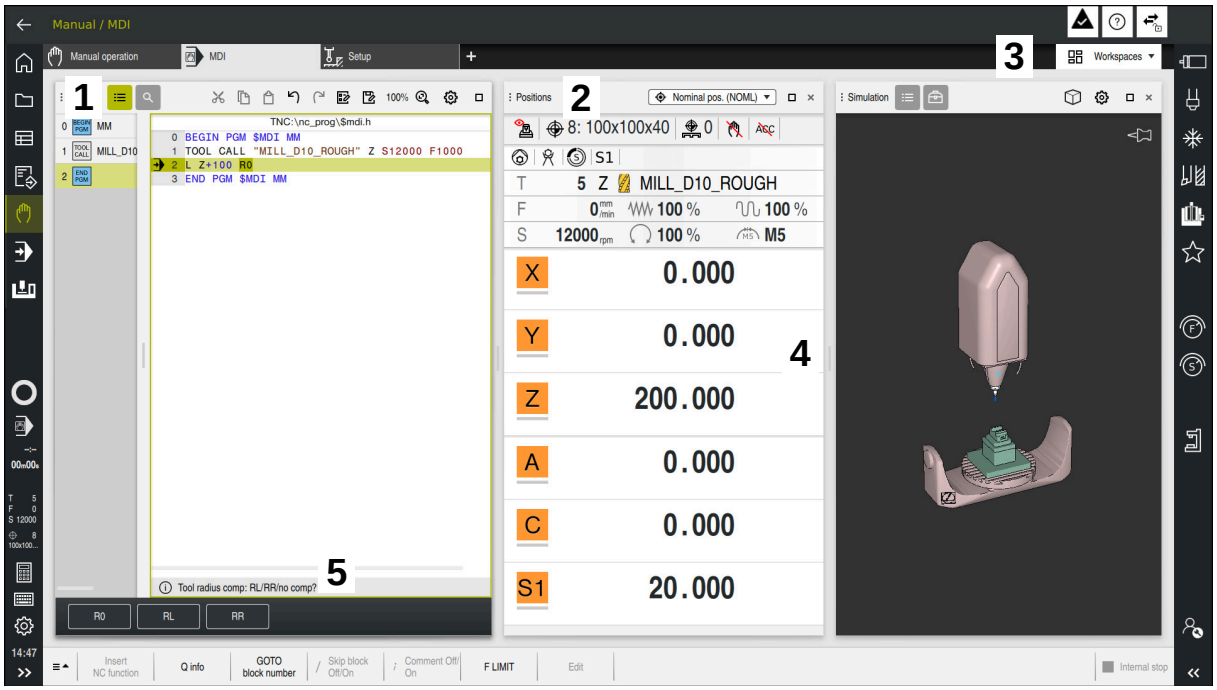
3.6 Overview of the operating modes

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 ■ The **Help** application ■ Applications for machine parameters	 Page 557 See the User's Manual for Programming and Testing Page 626
	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.	See the User's Manual for Programming and Testing
	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 442
	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	See the User's Manual for Programming and Testing
	The **Manual** operating mode contains the following applications: ■ The **Manual operation** application ■ The **MDI** Application ■ The **Setup** application ■ The **Move to ref. point** application ■ The **Retract** application You can move the tool away from the workpiece, for example after a power failure.	 Page 162 Page 369 Page 373 Page 157 Page 438
	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence. You also execute pallet tables in this operating mode.	Page 412
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	Page 545
	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.	Page 373
Job list In the **Job list** workspace, you edit and execute pallet tables.	See the User's Manual for Programming and Testing
Open File In the **Open File** workspace you select or create files, for example.	See the User's Manual for Programming and Testing
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.	See the User's Manual for Programming and Testing
Details In the **Details** workspace, the control displays information on the selected machine parameter or the last change you made.	Page 631
Document You can open files for viewing in the **Document** workspace, for example a technical drawing.	See the User's Manual for Programming and Testing
Settings In the **Settings** workspace, you can display and edit, if required, various settings of the control (e.g., set up the traverse limits). The **Settings** workspace is part of the **Settings** application.	Page 557
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 454
The **Form** for pallets In the **Form** workspace the control shows the contents of the pallet table for the selected row.	See the User's Manual for Programming and Testing

Workspace	Further information
Retract In the **Retract** workspace, you can disengage the tool after a power interruption.	Page 438
GPS (#44 / #1-06-1) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	Page 287
Desktop menu In the **Desktop menu** workspace, the control displays selected control and HEROS functions.	Page 100
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**.	See the User's Manual for Programming and Testing
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.	See the User's Manual for Programming and Testing
List In the **List** workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	Page 627
Positions In the **Positions** workspace, the control displays information about the status of various functions of the control and about current axis positions.	Page 119
Program The control displays the NC program in the **Program** workspace.	See the User's Manual for Programming and Testing
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.	Page 307
Referencing On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.	Page 157
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.	Page 545
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.	See the User's Manual for Programming and Testing

Workspace	Further information
Simulation In the **Simulation** workspace, the control shows the simulated or current movements, depending on the operating mode.	See the User's Manual for Programming and Testing
Simulation status In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	Page 146
Start/Login In the **Start/Login** workspace, the control shows the steps that are performed during startup.	Page 105
Status In the **Status** workspace, the control shows the status and values of individual functions.	Page 127
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 447
The **Table** for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	Page 627
Keyboard In the **Keyboard** workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 362
Overview In the **Overview** workspace, the control displays information on the status of individual functional safety (FS) safety functions.	Page 552

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 89

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

Description of function

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

> If there are deviations from the virtual keyboard, the table also indicates the corresponding keys on the virtual keyboard.
>
> **Further information:** "Virtual keyboard of the control bar", Page 362

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:** Programming and Testing User's Manual 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
? /	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:** "Connection settings", Page 610
	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:** Programming and Testing User's Manual
↻	Currently no function
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 365
CALC	Open and close the calculator **Further information:** Programming and Testing User's Manual
MOD	Open the **Settings** application **Further information:** "The Settings application", Page 557
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 53

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:** "The Manual operation application", Page 162
	Activate and deactivate the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic handwheel", Page 519
	Open the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 197
	Open the **MDI** application in the **Manual** operating mode **Further information:** "The MDI Application ", Page 369
	Open the **Program Run** operating mode in **Single Block** mode **Further information:** "The Program Run operating mode", Page 412
	Open the **Program Run** operating mode **Further information:** "The Program Run operating mode", Page 412
	Open the **Editor** operating mode **Further information:** Programming and Testing User's Manual
	While the NC program is running, open the **Simulation** workspace in the **Editor** operating mode **Further information:** Programming and Testing User's Manual

Keycaps for NC dialog

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

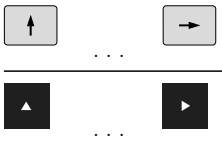
Key	Meaning
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode
CHF	Program a chamfer
L	Program a straight line segment
CR	Program a circular arc with radius entry
RND	Program a rounding arc
CT	Program a circular arc with tangential connection to the preceding contour element
CC	Program a circle center or pole
C	Program a circular arc with reference to the circle center
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
LBL CALL	Program a subprogram or a program section repeat
STOP	Program an intentional stop
TOOL DEF	Pre-select a tool in the NC program
TOOL CALL	Call the tool in the NC program
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)

Further information: Programming and Testing User's Manual

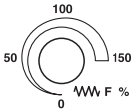
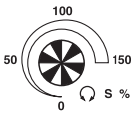
Keycaps for axis input and value input

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:** "Status overview on the TNC bar", Page 125 In the **Editor** operating mode and the **MDI** application, program a line **L** with the actual positions of all defined axes.
Q	In the **Editor** operating mode, open the **FN** folder in the **Insert NC function** window
FN	
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
	Increase or reduce the feed rate **Further information:** Programming and Testing User's Manual
	Increase or reduce the spindle speed **Further information:** Programming and Testing User's Manual

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**
	Call context-sensitive help **Further information:** "Context-sensitive help", Page 56
	Dynamic collision monitoring (DCM) is deactivated **Further information:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246

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:** "Automatic program start", Page 433
	**Window manager** Select active applications in the background (e.g., HEROS functions window)
	ITC is connected and **Enabling VNC focus** is set **Further information:** "The VNC Focus Settings area", Page 607 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.

  Further information: "Configuring the control's user interface", Page 632
- Full-text search

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

  Further information: "Adding and removing favorites", Page 101

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 84

  Further information: "Overview of workspaces", Page 86
- **Tools**

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

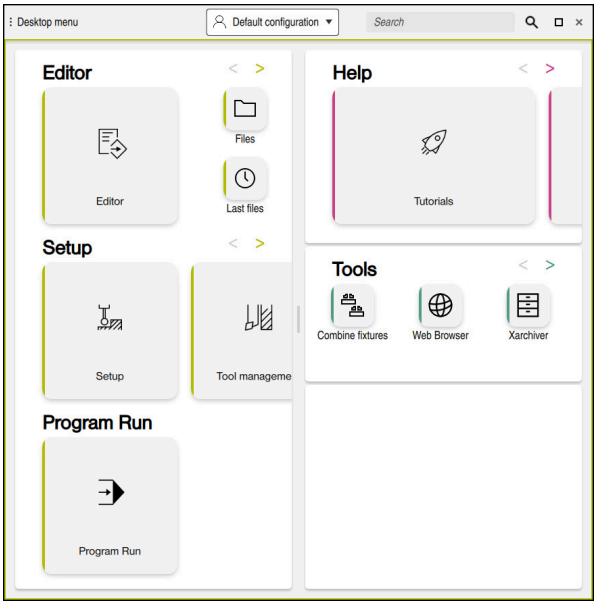
  Further information: "HEROS operating system", Page 661
- **Help**

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

  Further information: "User's Manual as integrated product aid: TNCguide", Page 53
- **Favorites**

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

  Further information: "Adding and removing favorites", Page 101



The **Desktop menu** workspace

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

Showing or hiding an area

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

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

> Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

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

- ▶ Use the full-text search
- ▶ 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

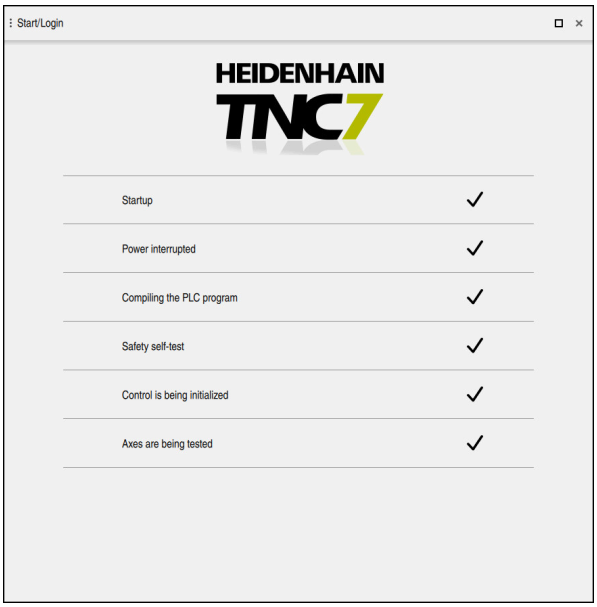
4.1 Chapter overview

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
- Setting up a tool
- Setting up a workpiece
- Machining 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: "The Referencing workspace", Page 157

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

  Further information: "The Manual operation application", Page 162

More detailed information

- Switching on and off
- Position encoders

  Further information: "Position encoders and reference marks", Page 173
- Axis reference run

4.3 Setting up a tool

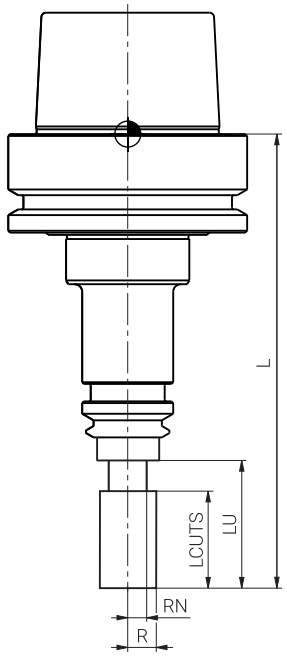
This chapter uses an example to show how to set up a tool.

To set up a tool, the following steps are necessary:

- Preparing the tools
- Opening the **Tool management** application and the **Form** workspace
- Inserting a tool in the tool management
- Defining the tool
- Entering the tool in the pocket table

You have to set up the tools to enable the control to position, simulate and compensate for the tools, for example.

4.3.1 Example tool

Tool	Parameter	Value
	**Name**	MILL_D12
	L	+120
	R	+6
	R2	+0
	TYP	**Roughing mill** (**MILL_R**)
	LCUTS	+20
	LU	28
	RN	4.8
	R-TIP	0
	T-ANGLE	+0
	CUTS	4

4.3.2 Preparing the tools

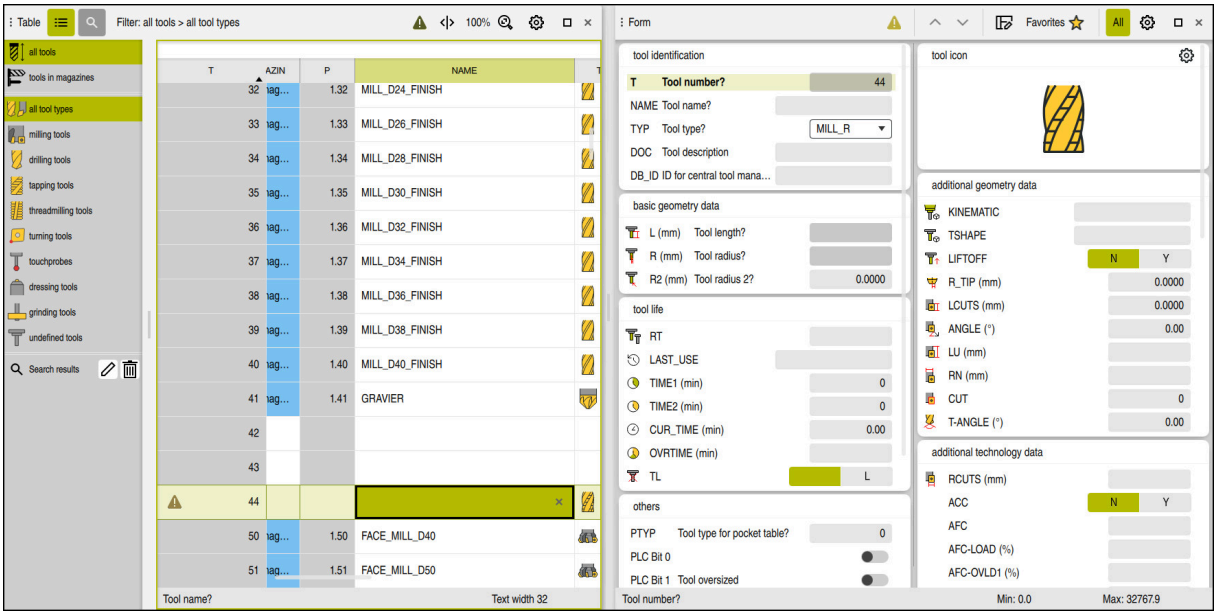
To prepare tools:

- ▶ Clamp the required tools in the tool holder
- ▶ Measure the tools (e.g., with a presetter)
- ▶ Write down the parameters
- > Now the tool is ready.

- You can measure tools with the tool touch probe of the control.
- Alternatively, the control makes it possible to measure the tool without a tool touch probe.

  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- You cannot and do not have to measure all parameters. Some values can be found in the tool catalog for example.

4.3.3 Opening the Tool management application and the Form workspace



The **Tool management** application with the **Table** and **Form** workspaces

You enter tools in the **Tool management** application of the **Tables** operating mode. You define tool parameters in the **Form** workspace. In the **Form** workspace, the control shows the required parameters of the selected tool.

To open the **Tool management** application and the **Form** workspace:

- ▶ Select the **Tables** operating mode
- > The control displays the **Tables** operating mode.
- ▶ Select **Tool management**
- > The control displays the **Tool management** application.
- ▶ In the application bar, select **Workspaces**
- ▶ Select **Form**
- > The control opens the **Form**Simulation workspace.

More detailed information

- **Tables** operating mode
  Further information: "The Tables operating mode", Page 442
- The **Form** workspace
  Further information: "The Form workspace for tables", Page 454
- Tool parameters
  Further information: "Tool parameters", Page 185
- Tool management
  Further information: "Tool management ", Page 197

4.3.4 Inserting a tool in the tool management

To insert a tool:

- Activate **Edit**

- Select **Insert tool**
- The control opens the **Insert tool** window.
- Select the tool type, for example **Roughing mill** (**MILL_R**)
- Enter the line number, if required

- Select **OK**
- The control inserts the new tool

More detailed information

- Tool types

  Further information: "Tool types", Page 194

4.3.5 Defining the tool

To define the tool with the measured parameters:

- Enter the tool name **NAME** (e.g., **MILL_D12**)
- Enter the tool length **L** (e.g., **120**)
- Enter the tool radius **R** (e.g., **6**)
- Enter the tooth **LCUTS** (e.g., **20**)
- Enter the usable length **LU** (e.g., **28**)
- Enter the neck radius **RN** (e.g., **4,8**)
- Enter the number of teeth **CUTS** (e.g., **4**)
- The control saves the tool with all entered parameters.

Based on the entered parameters, the control shows the tool correctly in the **Simulation** workspace.

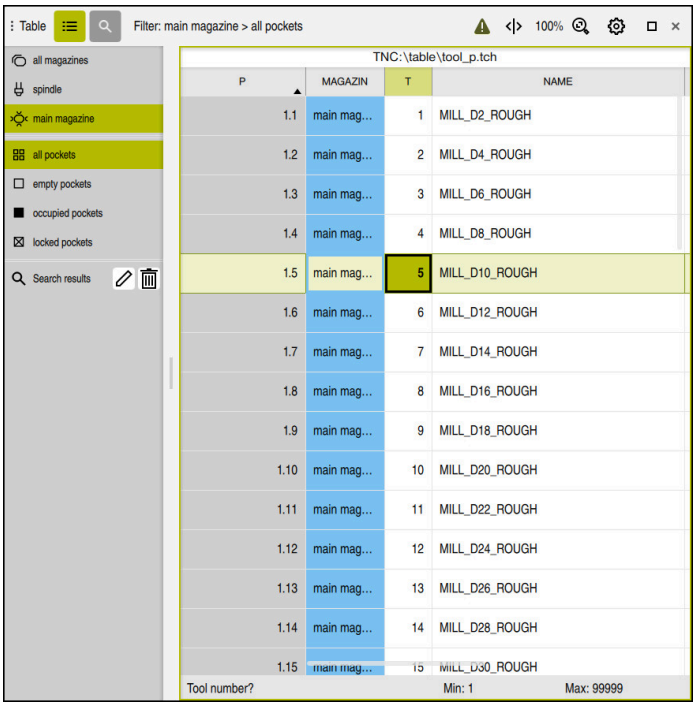
Further information: Programming and Testing User's Manual

To use the full performance range, you have to define further parameters, such as **KINEMATIC** (tool-carrier kinematics for collision monitoring). You need different parameters depending on the tool type.

Further information: "Tool parameters", Page 185

4.3.6 Entering the tool in the pocket table

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



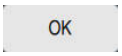
Pocket table application with opened **Table** workspace

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. You must add the tool to the pocket table in order for the control to be able to insert the tool. The tool must be stored in the machine.

To enter a tool in the pocket table:

- ▶ Select the **Pocket table** application
- > The control displays the **Pocket table** application.
- ▶ Enter the desired pocket number in the **Table** workspace
- ▶ Open the **Form** workspace

- ▶ Activate **Edit**
- ▶ Select the tool number **T**
- > The control opens the **Tool selection** window.
- ▶ Select the tool you want to use



- ▶ Select **OK**
- > The control inserts the tool in the pocket table.
- ▶ Define additional parameters if necessary (e.g., to reserve a pocket)

More detailed information

- Pocket table
  Further information: "Pocket table tool_p.tch", Page 494

4.4 Setting up a workpiece

4.4.1 Selecting an operating mode

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

To select the **Manual** operating mode:

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

More detailed information

- Operating mode: **Manual**
  Further information: "Overview of the operating modes", Page 84

4.4.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

More detailed information

- The **Probing function** workspace
  Further information: "Touch probe functions in the Manual operating mode", Page 373

4.4.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

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

To insert a workpiece touch probe:

- Select **T**
- Enter the tool number of the workpiece touch probe (e.g., **600**)

- Press the **NC Start** key
- The controls inserts the workpiece touch probe.

Setting a workpiece preset

To set a workpiece preset at a corner:

- Select the **Setup** application

- Select **Intersection point (P)**
- > The control opens the probing cycle.

- Select **Change preset**, if necessary
- > The control opens the **Change the preset** window.
- Choose another row in the preset table if necessary.
- > The control highlights the selected line in green.

- Select **Apply** , if appropriate
- Manually position the touch probe near the first touch point of the first workpiece edge

- In the **Choose the probing direction** area, select the direction of probing (e.g., **Y+**)

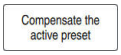
- Press the **NC Start** key
- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the first workpiece edge

- Press the **NC Start** key
- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the first touch point of the second workpiece edge

- In the **Choose the probing direction** area, select the direction of probing (e.g., **X+**)

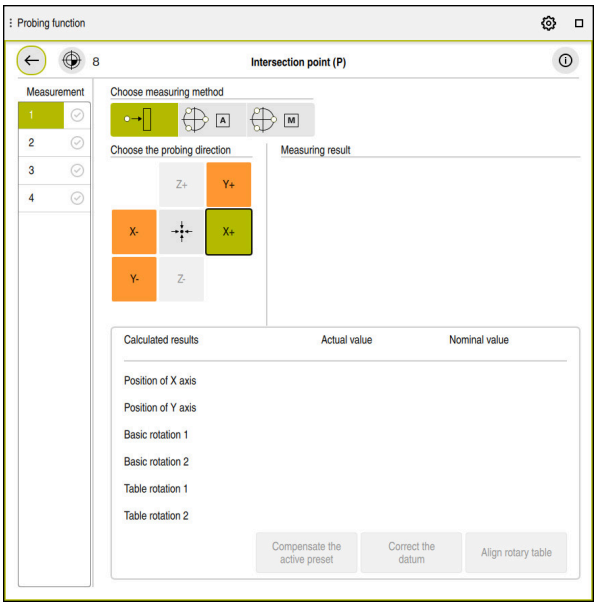
- Press the **NC Start** key
- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the second workpiece edge

- Press the **NC Start** key
- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- > The control then displays the coordinates of the determined corner point in the **Measuring result** area.



- Select **Compensate the active preset**
- > The control applies the calculated results to the workpiece preset.

- Select **Exit probing**
- > The control closes the probing cycle.



The **Probing function** workspace with an open manual probing function

More detailed information

- The **Probing function** workspace

  Further information: "Touch probe functions in the Manual operating mode", Page 373
- Presets in the machine

  Further information: "Presets in the machine", Page 174
- Preset management

  Further information: "Preset management", Page 230
- Tool change in the **Manual operation** application

  Further information: "The Manual operation application", Page 162

4.5 Machining a workpiece

4.5.1 Selecting an operating mode

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

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

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

More detailed information

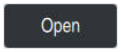
- The **Program Run** operating mode
  Further information: "The Program Run operating mode", Page 412

4.5.2 Opening an NC program

To open an NC program:

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

- Select an NC program



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

More detailed information

- The **Open File** workspace
  Further information: Programming and Testing User's Manual

4.5.3 Starting an NC program

To start an NC program:

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

4.6 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

- Select **Shut down**

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

- 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

Status displays

5.1 Overview

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

The control offer the following status displays:

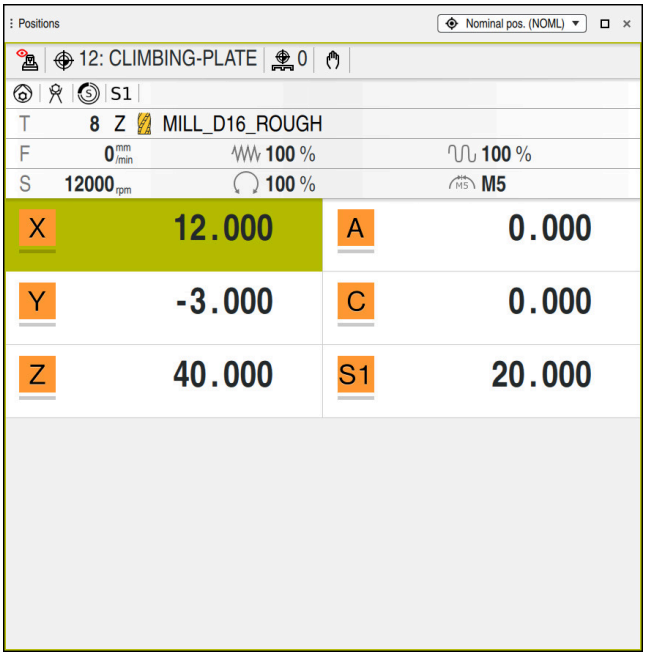
- General status display and position display in the **Positions** workspace
  Further information: "The Positions workspace", Page 119
- Status overview on the TNC bar
  Further information: "Status overview on the TNC bar", Page 125
- Additional status displays for specific areas in the **Status** workspace
  Further information: "The Status workspace", Page 127
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
  Further information: "The Simulation status workspace", Page 146

5.2 The Positions workspace

Application

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

Description of function



The **Positions** workspace with general status display

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

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

Further information: "Overview of the operating modes", Page 84

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

- Icons of active and inactive functions (e.g., Dynamic Collision Monitoring DCM (#40 / #5-03-1))
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"

  Further information: "Test status of the axes", Page 554

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.

Axis display and position display

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

Icon	Meaning
IST	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. **Further information:** "Position displays", Page 149
X	Axes The X axis is selected. You can move the selected axis.
m	The auxiliary axis **m** is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. **Further information:** "Definition", Page 124
?	The axis is not referenced.
⚠	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 555
Δ	The axis is moving the distance-to-go shown next to the symbol.
⊣⊢	The axis is clamped.
	You can move the axis with the handwheel.
	You cannot move the axis with the handwheel. Refer to your machine manual. The machine manufacturer defines which axes you can move with the handwheel.
F_{SOS} 0 mm/min	Feed status when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 552
S_{SOS} 0 U/min	Spindle status when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 552

Presets and technology values

Icon	Meaning
	Number and comment of the active workpiece preset The number corresponds to the active row number of the preset table. The comment corresponds to the content of the **DOC** column. **Further information:** "Preset management", Page 230
	Number of the active pallet preset The number corresponds to the active row number in the pallet preset table. **Further information:** Programming and Testing User's Manual
T	In the **T** area, the control shows the following information: ■ Number of the active tool ■ Tool axis of the active tool ■ Symbol of the defined tool type ■ Name of the active tool
F	In the **F** area, the control shows the following information: ■ Active feed rate in mm/min You can program the feed rate in various units of measurement. The control always converts the programmed feed rate in this display to mm/min. ■ If **M136** is active: active feed rate in mm/rev **Further information:** Programming and Testing User's Manual ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 95 If a feed-rate limitation has been activated with the **F LIMIT** button, the area is labeled **F LIMIT** instead of **F**. The control displays the text **F LIMIT** and the feed-rate value in orange. **Further information:** "Feed rate limit F LIMIT", Page 417
S	In the **S** area, the control shows the following information: ■ Active shaft speed in rpm If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed. ■ Setting of the spindle potentiometer in percent ■ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The **Manual traverse** function is active.
	The **Manual traverse** function is not active. **Further information:** "The Program Run operating mode", Page 412
	**RL** tool radius compensation is active. **Further information:** Programming and Testing User's Manual
	**RR** tool radius compensation is active. **Further information:** Programming and Testing User's Manual These symbols are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 424
	**R+** tool radius compensation is active. **Further information:** Programming and Testing User's Manual
	**R-** tool radius compensation is active. **Further information:** Programming and Testing User's Manual These symbols are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 424
	3D tool compensation is active (#9 / #4-01-1). **Further information:** Programming and Testing User's Manual This symbol is transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 424
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 232
	The basic rotation will be taken into account while moving the axes. **Further information:** "The Basic rotation selection item", Page 242
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 232

Icon	Meaning
	The tilted working plane will be taken into account while moving the axes. **Further information:** Programming and Testing User's Manual **Further information:** "The 3D ROT selection item", Page 242
	The **Tool axis** function is active. **Further information:** "The Tool axis selection item", Page 242
	Either the **TRANS MIRROR** function or Cycle **8 MIRRORING** is active. The axes programmed in the function or cycle are mirrored and moved. **Further information:** User's Manual for Machining Cycles **Further information:** Programming and Testing User's Manual
	The pulsing spindle speed function **S-PULSE** is active. **Further information:** Programming and Testing User's Manual
	The **PARAXCOMP DISPLAY** function is active.
	The **PARAXCOMP MOVE** function is active. **Further information:** Programming and Testing User's Manual
	If one of the following NC functions is active, then the control displays this symbol: ■ **CYLINDER SURFACE** (#8 / #1-01-1) ■ **FUNCTION PARAX MODE** ■ **POLARKIN** (#8 / #1-01-1) This icon might be superimposed on the icons for **FUNCTION PARAX COMP DISPLAY** and **FUNCTION PARAX COMP MOVE**. **Further information:** Programming and Testing User's Manual
TCPM	The function **M128** or **FUNCTION TCPM** is active (#9 / #4-01-1). **Further information:** Programming and Testing User's Manual
	Turning mode **FUNCTION MODE TURN** is active (#50 / #4-03-1). **Further information:** Programming and Testing User's Manual
	The cylindrical grinding mode **FUNCTION MODE GRIND** is active (#156 / #4-04-1). **Further information:** Programming and Testing User's Manual
	Dressing mode is active (#156 / #4-04-1). **Further information:** Programming and Testing User's Manual

Icon	Meaning
	The Dynamic Collision Monitoring function (DCM) is active (#40 / #5-03-1).
	The Dynamic Collision Monitoring function (DCM) is not active (#40 / #5-03-1). **Further information:** "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
	The Dynamic Collision Monitoring function (DCM) is active with a reduced minimum distance (#140 / #5-03-2). **Further information:** Programming and Testing User's Manual
	The Adaptive Feed Control function (AFC) is active in teach-in cut mode (#45 / #2-31-1).
AFC	The Adaptive Feed Control function (AFC) is active in closed-loop mode (#45 / #2-31-1). **Further information:** "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276
ACC	The Active Chatter Control function (ACC) is active (#145 / #2-30-1). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 286
	The Global program settings function (GPS) function is active (#44 / #1-06-1). **Further information:** "Global Program Settings GPS (#44 / #1-06-1)", Page 287
	The Process monitoring function is active (#168 / #5-01-1). **Further information:** "Process monitoring (#168 / #5-01-1)", Page 302

In the optional machine parameter **iconPrioList** (no. 100813), you can change the sequence in which the control displays these symbols. The symbol for Dynamic Collision Monitoring (DCM) (#40 / #5-03-1) is always visible and cannot be configured.

Definition

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

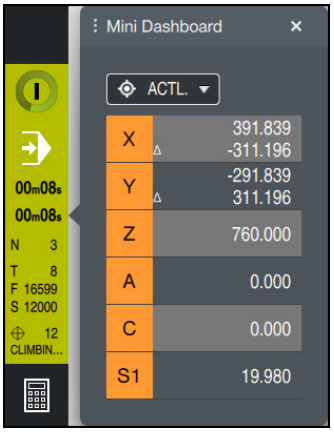
5.3 Status overview on the TNC bar

Application

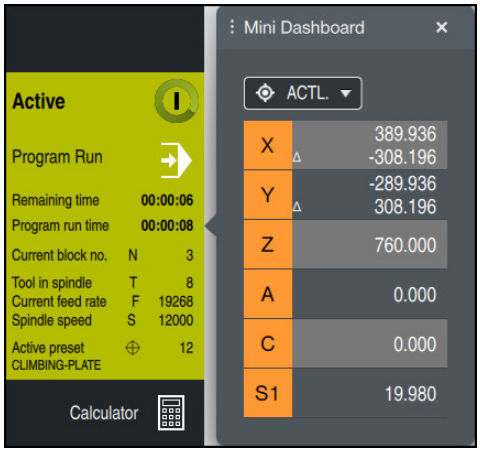
On the TNC bar, the control shows a status overview with the execution status, the current technology values, and the axis positions.

Description of function

General information



Status overview of the collapsed TNC bar with opened position display



Status overview of the expanded TNC bar with opened position display

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

- **Control-in-operation**: current machining status
  Further information: "Definition", Page 126
- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time
- Active tool
- Active feed rate
- Current spindle speed
- Number and comment of the active workpiece preset
- Position display

If the TNC bar is expanded, then the control displays the run times in the default format hh:mm:ss (e.g., 01:10:30).

If the TNC bar is collapsed, then the control displays the run times in the following formats:

- Minutes and seconds for run times of less than an hour (e.g., 10min30s)
- Hours and minutes for run times of one hour or longer (e.g., 01h04min)

Further information: "Display of the program run time", Page 147

Position display

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

Further information: "The Positions workspace", Page 119

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

Press the **actual position capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard. If you select the value of an axis while editing an NC block, then the control inserts the value into the NC block.

Further information: Programming and Testing User's Manual

Definition

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

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

Further information: "Interrupting, stopping or canceling program run", Page 418

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

5.4 The Status workspace

Application

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

Description of function

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

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

Further information: "Overview of the operating modes", Page 84

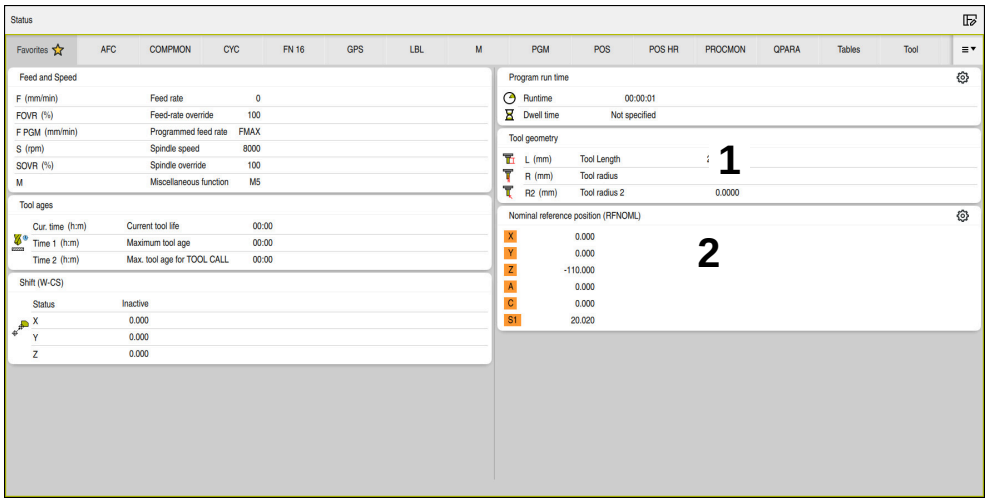
Icons

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

Icon	Meaning
	**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
	**Settings** Some areas have their own settings. Use this icon to customize the contents of the area (e.g., by defining the variable range to be displayed).
	**Favorite** **Further information:** "The Favorites tab", Page 128
	**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 456 ■ Area In the **Favorites** view you can add another area.
	Remove The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

The Favorites tab

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



The **Favorites** tab

1 Area

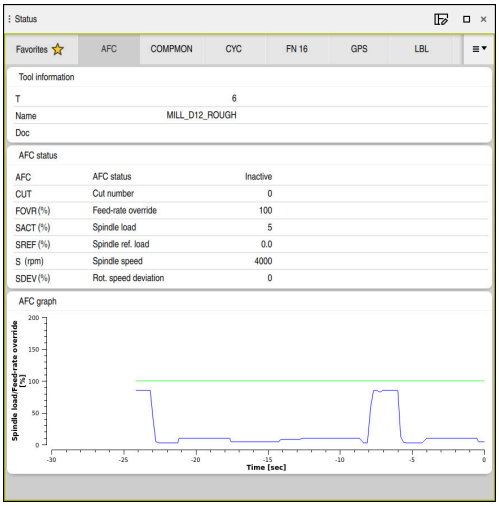
2 Contents

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

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

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

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276



AFC tab

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Comment about the tool from the tool management
AFC status	■ **AFC** If AFC is being used to control the feed rate, then **Control** is displayed in this area. If the control is not controlling the feed rate, then **Inactive** is displayed in this area. ■ **CUT** Counts the quantity of cuts that have been performed with **FUNCTION AFC CUT BEGIN**, starting from zero. ■ **FOVR (%)** Active factor of the feed-rate potentiometer in percent ■ **SACT (%)** Current spindle load in percent ■ **SREF (%)** Reference load of the spindle in percent Define the reference load of the spindle in the syntax element **LOAD** of the **FUNCTION AFC CUT BEGIN** function. **Further information:** "NC functions for AFC (#45 / #2-31-1)", Page 279 ■ **S (rpm)** Spindle shaft speed in rpm ■ **SDEV (%)** Current deviation of the speed in percent

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

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

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

Further information: Programming and Testing User's Manual

Refer to your machine manual.

The machine manufacturer defines the monitoring functions and the contents of the **COMPMON** tab.

The **COMPMON** tab with configured spindle speed monitoring

The machine manufacturer can define up to four ranges and their content on the **COMPMON** tab.

The machine manufacturer can select the following values for each component:

- Status

  The control shows the individual status of up to five components and the combined results of all components as a colored bar.

  - Green: component is the range that is defined as safe
  - Yellow: component in the warning zone
  - Red: component is overloaded

  If a component cannot be monitored, then the control displays the status as gray. A component cannot be monitored, for example, if configurations are missing or faulty.

- Graph of current monitoring

  The control displays the graph either as a resulting value relative to the defined boundaries or as an absolute display of the signal.
- Histogram

  The control shows a graphical evaluation of previous monitoring sessions.

- You can zoom in or out of the graph horizontally by scrolling or dragging.
- You can shift the graph by swiping or while pulling with the left mouse button pressed.

In the **Program Run** operating mode, the control monitors the components only during machining. If no machining is active, then the control displays the value of the most recent program run.

CYC tab

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

Area	Contents
Active cycle definition	When you use the **CYCL DEF** function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLERANCE	■ **Status** Shows whether Cycle **32 TOLERANCE** is active or inactive ■ Values of Cycle **32 TOLERANCE** ■ Values from the machine manufacturer for path and angle tolerance, such as predefined machine-specific roughing or finishing filters ■ Values of Cycle **32** limited by Dynamic Collision Monitoring (DCM) **TOLERANCE** (#40 / #5-03-1) If a value is limited by DCM, then the control displays the the text **DCM limited** following the value.

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

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

The FN 16 tab

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

Further information: Programming and Testing User's Manual

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

The GPS tab (#44 / #1-06-1)

The control displays information on the Global Program Settings (GPS) (#44 / #1-06-1) on the **GPS** tab.

Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287

Area	Contents
Additive offset (M-CS)	■ **Status** The **Status** shows whether a function is active or inactive. A function can be active even if its values are zero. ■ **A (°)** **Additive offset (M-CS)** in the A axis The **Additive offset (M-CS)** function is also available for the other rotary axes **B (°)** and **C (°)**.
Additive basic rotat. (W-CS)	■ **Status** ■ **(°)** The **Additive basic rotat. (W-CS)** function is active in the workpiece coordinate system **W-CS**. Entries are in degrees. **Further information:** "Workpiece coordinate system W-CS", Page 220
Shift (W-CS)	■ **Status** ■ **X** **Shift (W-CS)** in the X axis The **Shift (W-CS)** function is also available for the other linear axes **Y** and **Z**.
Mirroring (W-CS)	■ **Status** ■ **X** **Mirroring (W-CS)** in the X axis The **Mirroring (W-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.

Area	Contents
Rotation (WPL-CS)	■ **Status** ■ **(°)** **Rotation (WPL-CS)** in degrees The **Rotation (WPL-CS)** function is active in the working plane coordinate system **WPL-CS**. Entries are in degrees. **Further information:** "Working plane coordinate system WPL-CS", Page 222
Shift (mW-CS)	■ **Status** ■ **X** **Shift (mW-CS)** in the X axis The **Shift (mW-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.
Handwheel superimp.	■ **Status** ■ **Coordinate system** This area contains the selected coordinate system for **Handwheel superimp.**, such as the machine coordinate system **M-CS**. ■ **X** ■ **Y** ■ **Z** ■ **A (°)** ■ **B (°)** ■ **C (°)** ■ **VT**
Feed rate factor	If the **Feed rate factor** function is active, the control displays the defined percentage in this field. If the **Feed rate factor** function is not active, the control displays **100.00 %** in this field.

The LBL tab

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

Further information: Programming and Testing User's Manual

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

The M tab

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

Further information: Programming and Testing User's Manual

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

The PGM tab

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

Area	Contents
Parts counter	**Quantity** Actual value and nominal value of the parts counter defined with the **FUNCTION COUNT** function **Further information:** Programming and Testing User's Manual The **Settings** icon lets you open the **Counter settings** window, in which you can check and, if needed, edit the current count and the target value for the counter. You cannot edit these values while the control is executing an NC program. With the optional machine parameter **userPermission** (no. 129101), the machine manufacturer defines which counter settings you can change.
Program run time	■ **Runtime** Run time of the NC program in the format hh:mm:ss ■ **Dwell time** Countdown of the waiting time in seconds from the following functions: ■ **FUNCTION DWELL** ■ Cycle **9 DWELL TIME** ■ Parameter **Q210 DWELL TIME AT TOP** ■ Parameter **Q211 DWELL TIME AT DEPTH** ■ Parameter **Q255 DWELL TIME** **Further information:** "Display of the program run time", Page 147
Programs called	Path of the main program as well as called NC programs including the path

Area	Contents
Pole/circle center	Programmed axes and values of the circle center point **CC**
Radius compensation	Programmed tool radius compensation
Program run options	Active breakpoints in connection with the override controller **Further information:** "Override controller", Page 537

The POS tab

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

Area	Contents
Position display (e.g., **Actual reference position (RFACTL)**)	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ **Nominal pos. (NOML)** ■ **Actual pos. (ACT)** ■ **Nominal reference position (RFNOML)** ■ **Actual reference position (RFACTL)** ■ **Servo lag (LAG)** ■ **Handwheel superimposed (M118)** **Further information:** "Position displays", Page 149
Feed and Speed	■ Active **Feed** in mm/min If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the **F LIMIT** button, the control displays **LIMIT** in square brackets. **Further information:** "Feed rate limit F LIMIT", Page 417 If the feed rate is limited by means of functional safety (FS), then the control displays the active safety function in brackets. **Further information:** "Safety functions", Page 551 ■ Active **Feed-rate override** in % ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min If **M136** is active: active feed rate in mm/rev **Further information:** Programming and Testing User's Manual ■ Active **Spindle speed** in rpm ■ Active **Spindle override** in % ■ Active **Miscellaneous function** in reference to the spindle, such as **M3**  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.

Area	Contents
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** Programming and Testing User's Manual If axis angles are active, the control displays in this area only the values of the physically present axes. Defined values in the **3-D rotation** window **Further information:** "The 3D ROT selection item", Page 242
OEM transformation	The machine manufacturer can define an OEM transformation for special turning kinematics. **Further information:** "Definitions", Page 145
Basic transformations	In this area the control shows the values of the active workpiece preset and active transformations in linear and rotary axes, such as a transformation in the X axis with the function **TRANS DATUM**. **Further information:** "Preset management", Page 230
Special turning transformations	Transformations relevant for turning operations (#50 / #4-03-1), such as the defined **precession angle** from the following sources: ■ Defined by the machine manufacturer ■ Cycle **800 ADJUST XZ SYSTEM** ■ Cycle **801 RESET ROTARY COORDINATE SYSTEM** ■ Cycle **880 GEAR HOBBING**
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1 Traverse ranges are machine-specific. If no traverse range is active, then **Traverse range not defined** is displayed in this area.
Active kinemat.	Name of the active machine kinematics

The POS HR tab

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

Area	Contents
Coordinate system	■ **Machine (M-CS)** If you use **M118**, handwheel superimpositioning is always effective in the machine coordinate system **M-CS**. **Further information:** Programming and Testing User's Manual With the Global Program Settings (GPS) (#44 / #1-06-1), the coordinate system can be selected. **Further information:** "Global Program Settings GPS (#44 / #1-06-1)", Page 287
Handwheel superimp.	■ **Max. val.** Maximum value of the individual axes, programmed in **M118** or in the **GPS** workspace (#44 / #1-06-1) ■ **Actl.val.** Current superimpositioning

The PROCMON (#168 / #5-01-1) tab

On the **PROCMON** tab, the control displays information about process monitoring.

Use the **PROCMON** tab after you have finished setting up process monitoring and no longer need to make any adjustments. Unlike the **Process Monitoring** workspace, the **PROCMON** tab offers a compact overview of the current machining operation.

Further information: "Process monitoring (#168 / #5-01-1)", Page 302

The control displays the **PROCMON** tab only in the **Program Run** operating mode.

Area	Contents
Monitoring tasks for process monitoring	■ Current status of all active monitoring tasks ■ Combined graph of all monitoring tasks, showing the largest deviations
Overview of process monitoring	■ **Status** inactive, active or within a monitoring section ■ **Main program** Path of the active NC program ■ **Monitoring sections** (quantity) ■ **Current monitoring section** (name) ■ **Duration of monitoring section** ■ **Progress of monitoring section** ■ **Visualization on workpiece** Monitoring task whose status is shown on the workpiece as a heat map ■ **Events** (quantity) ■ **Pallet** Information about whether the NC program is part of pallet machining

The QPARA tab

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

Further information: Programming and Testing User's Manual

Using the **Parameter list** and **Variable list** windows, you can define which variables the control shows in the areas. Up to 22 variables can be displayed in each area.

Further information: "Defining the contents of the QPARA tab", Page 151

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

The Tables tab

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

Area	Contents
Active tables	In this area the control shows the path for the following active tables: ■ Tool table ■ Turning-tool table (#50 / #4-03-1) ■ Preset table ■ Datum table ■ Pocket table ■ Touch-probe table ■ Grinding tool table (#156 / #4-04-1) ■ Dressing tool table (#156 / #4-04-1)

The TRANS tab

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

Area	Contents
Active datum	■ Path of the selected datum table ■ Row number of the selected datum table ■ **DOC** Contents of the **DOC** column of the datum table
Active datum shift	Datum shift that was defined with the **TRANS DATUM** function **Further information:** Programming and Testing User's Manual
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** Programming and Testing User's Manual **Further information:** User's Manual for Machining Cycles

Area	Contents
Active angle of rotation	Rotation angle defined with either the **TRANS ROTATION** function or Cycle **10 ROTATION** **Further information:** Programming and Testing User's Manual **Further information:** User's Manual for Machining Cycles
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** Programming and Testing User's Manual
Center of scaling	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** User's Manual for Machining Cycles
Active scaling factors	Scaling factors that were defined for the individual linear axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** Programming and Testing User's Manual **Further information:** User's Manual for Machining Cycles
Shift (WPL-CS)	Active shift in the working plane coordinate system **WPL-CS** using the following function: ■ **FUNCTION CORRDATA** ■ **FUNCTION TURNDATA CORR** (#50 / #4-03-1) **Further information:** Programming and Testing User's Manual
Table	■ Path of the selected compensation table ***.wco** ■ Row number of the selected compensation table ***.wco** ■ Content of the **DOC** column of the active row **Further information:** Programming and Testing User's Manual

The TT tab

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

Further information: "Hardware enhancements", Page 80

Area	Contents
TT: tool measurement	■ **T** Tool number ■ **Name** Tool name ■ **Measuring method** Selected measurement method for tool measurement (e.g., **Length**) ■ **Min (mm)** When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge. When measuring turning tools (#50 / #4-03-1), the control shows the smallest measured tilt angle in this area. The value of the angle can be negative. **Further information:** "Definitions", Page 145 ■ **Max (mm)** When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge. When measuring turning tools, in this area the control shows the greatest measured tipping angle. The value of the angle can also be negative. ■ **DYN Rotation (mm)** When measuring milling cutters with a rotating spindle, the control shows values in this area. When measuring turning tools, the value **DYN ROTATION** describes the tolerance for the tipping angle. If the tolerance for the tipping angle is exceeded during calibration, the control marks the affected value in the **MIN** or **MAX** fields with an *****. ⓘ In the optional machine parameter **tippingTolerance** (no. 114206) you define the tipping angle tolerance. The control will determine the tipping angle automatically only if a tolerance is defined.
TT: measurement of individual teeth	**Number** List of the measurements performed and the measured values of the individual cutting edges

The Tool tab

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

Further information: "Tool types", Page 194

Contents for dressing, milling and grinding tools (#156 / #4-04-1)

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Note on the tool
Tool geometry	■ **L** Tool length ■ **R** Tool radius ■ **R2** Corner radius of the tool
Tool allowances	■ **DL** Delta value for the tool length ■ **DR** Delta value for the tool radius ■ **DR2** Delta value for the corner radius of the tool With **Program**, the control displays the values from a tool call with **TOOL CALL** or from a tool compensation with a compensation table ***.tcs**. **Further information:** Programming and Testing User's Manual With **Table**, the control displays the values from the tool management. **Further information:** "Tool management ", Page 197
Tool ages	■ **Cur. time (h:m)** Time in hours and minutes the tool has been engaged ■ **Time 1 (h:m)** Service life of the tool ■ **Time 2 (h:m)** Maximum service life at tool call
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool
Tool type	■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**)

Deviating contents for turning tools (#50 / #4-03-1)

Area	Contents
Tool geometry	■ **ZL (mm)** Tool length in Z direction ■ **XL (mm)** Tool length in X direction ■ **RS (mm)** Cutter radius ■ **YL (mm)** Tool length in Y direction
Tool allowances	■ **DZL (mm)** Delta value in Z direction ■ **DXL (mm)** Delta value in X direction ■ **DRS (mm)** Delta value for the cutter radius ■ **DCW (mm)** Delta value for the width of the recessing tool ■ **WPL-DX-DIAM (mm)** Delta value for the workpiece diameter with respect to the working plane coordinate system **WPL-CS** Only if the **WPL-DX-DIAM** column has been defined in the turning-tool table **Further information:** "Working plane coordinate system WPL-CS", Page 222 ■ **WPL-DZL (mm)** Delta value for the workpiece length with respect to the working plane coordinate system **WPL-CS** Only if the **WPL-DZL** column has been defined in the turning-tool table **Further information:** "Working plane coordinate system WPL-CS", Page 222
Tool type	■ **Tool Axis** ■ **TO** Tool orientation ■ **Type** Tool type (e.g., **TURN**)

Definitions

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

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

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

5.5 The Simulation status workspace

Application

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

Description of function

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

- **Favorites**
  Further information: "The Favorites tab", Page 128
- **CYC**
  Further information: "CYC tab", Page 132
- **FN 16**
  Further information: "The FN 16 tab", Page 133
- **LBL**
  Further information: "The LBL tab", Page 134
- **M**
  Further information: "The M tab", Page 135
- **PGM**
  Further information: "The PGM tab", Page 135
- **POS**
  Further information: "The POS tab", Page 137
- **QPARA**
  Further information: "The QPARA tab", Page 140
- **Tables**
  Further information: "The Tables tab", Page 140
- **TRANS**
  Further information: "The TRANS tab", Page 140
- **TT**
  Further information: "The TT tab", Page 142
- **Tool**
  Further information: "The Tool tab", Page 143

Note

The control displays the current counter value and the defined target quantity only on the **PGM** tab of the **Status** workspace.

Further information: "The PGM tab", Page 135

Further information: Programming and Testing User's Manual

5.6 Display of the program run time

Application

The control calculates the duration of all traverse movements and displays them together as the **Program run time**. The control takes traversing movements and dwell times into account.

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

Description of function

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

- **PGM** tab of the **Status** workspace
  Further information: "The PGM tab", Page 135
- Status overview on the TNC bar
  Further information: "Status overview on the TNC bar", Page 125
- **PGM** tab of the **Simulation status** workspace
- The **Simulation** workspace in the **Editor** operating mode
  Further information: Programming and Testing User's Manual

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

The control opens a selection menu with the following functions:

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

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

The following functions reset the program run time:

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

Remaining run time of the NC program

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

Further information: "Tool usage test", Page 208

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

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

The following functions reset the remaining run time:

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

Notes

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

Definition

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

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

Further information: "Interrupting, stopping or canceling program run", Page 418

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

5.7 Position displays

Application

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

Description of function

The control has position displays in the following areas:

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

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

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	This mode shows the value of the currently calculated target position in the input coordinate system **I-CS**. When the machine moves the axes, the control compares the coordinates of the measured actual position with the calculated nominal position in predefined time intervals. The nominal position is the position at which the axes should be located at the time of comparison, based on the calculation. The **Nominal pos. (NOML)** and **Actual pos. (ACT)** modes differ solely with regard to the servo lag.
Actual pos. (ACT)	This mode shows the currently measured tool position in the input coordinate system **I-CS**. The actual position is the measured position of the axes, as determined by encoders at the time of comparison.
Nominal reference position (RFNOML)	This mode shows the calculated target position in the machine coordinate system **M-CS**. The **Nominal reference position (RFNOML)** and **Actual reference position (RFACTL)** modes differ solely with regard to the servo lag.
Actual reference position (RFACTL)	This mode shows the currently measured tool position in the machine coordinate system **M-CS**.
Servo lag (LAG)	This mode shows the difference between the calculated nominal position and the measured actual position. The control determines the difference in predefined time intervals.
Handwheel superimposed (M118)	This mode shows the values that you move using the **M118** miscellaneous function. **Further information:** Programming and Testing User's Manual

Refer to your machine manual.

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

5.7.1 Switching the position display mode

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

- Select the **POS** tab

- Select **Settings** in the position display area
- Select the desired mode for the position display (e.g., **Actual pos. (ACT)**)

> The control displays the positions in the selected mode.

Notes

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

  Further information: "Axis display and position display", Page 120

5.8 Defining the contents of the QPARA tab

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

Further information: "The QPARA tab", Page 140

To define the contents of the **QPARA** tab:

- Select the **QPARA** tab

- Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.
- Enter the number or name (e.g., **1,3,200-208**)

- Press **OK**
- > The control displays the values of the defined variables.

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

6

Powering on and off

6.1 Powering on

Application

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

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

Related topics

- Absolute and incremental position encoders
  Further information: "Position encoders and reference marks", Page 173

Description of function

DANGER

Caution: hazard to the user!

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

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

Power-on of the control begins with the power supply.
After booting, the controls checks the machine status, e.g.:

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

If the control registers an error during or after booting, it issues an error message.
The following step differs depending on position encoders on the machine:

- Absolute position encoders
  If the machine has absolute position encoders, the control opens the **Start/Login** application after power-on.
- Incremental position encoders
  If the machine has incremental position encoders, you must traverse the reference points in the **Move to ref. point** application. Once all axes have been referenced, the control is in the **Manual operation** application.
  Further information: "The Referencing workspace", Page 157
  Further information: "The Manual operation application", Page 162

6.1.1 Powering the machine and the control on

To switch the machine on:

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

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

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

  Further information: "The Referencing workspace", Page 157

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

  Further information: "The Manual operation application", Page 162

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

Further information: "The Functional safety application", Page 552

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

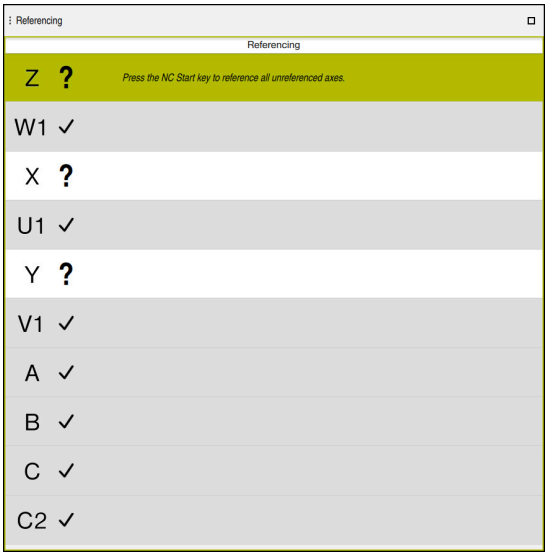
6.2 The Referencing workspace

Application

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

Description of function

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



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

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

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

6.2.1 Axis reference run

To reference the axes in the prescribed sequence:

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

To reference the axes in any sequence:

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

Notes

NOTICE

Danger of collision!

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

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

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

Notes about traversing reference points in a tilted working plane

If the function **Tilt working plane** (#8 / #1-01-1) was active before the control was shut down, then the control automatically activates the function after restarting. This means that movements via the axis keys take place in the tilted working plane.

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

Further information: Programming and Testing User's Manual

6.3 Powering off

Application

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

Description of function

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

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

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

6.3.1 Shutting down the control and powering-off the machine

To power-off the machine:

- Select the **Home** operating mode

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

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

Notes

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss regardless of the control's status!

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

- Different machines have different power-off procedures.
  Refer to your machine manual.
- Applications that are active on the control might delay the shutdown, such as a connection to **Remote Desktop Manager** (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

7

Manual Operation

7.1 The Manual operation application

Application

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

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 164
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 165

Description of function

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

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

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

Button	Meaning
Handwheel	The control displays this toggle switch if a handwheel is configured for the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic handwheel", Page 519
M	Define a miscellaneous function **M** or use the selection menu to choose one and activate it with the **NC start** key. **Further information:** Programming and Testing User's Manual The machine manufacturer uses the optional machine parameter **forbidManual** (no. 103917) to define which miscellaneous functions are allowed in the **Manual operation** application and are available in the selection menu.
S	Define the spindle speed **S**, activate it with the **NC start** key, and also switch on the spindle. **Further information:** Programming and Testing User's Manual
F	Define the feed rate **F** and activate it with the **OK** button. **Further information:** Programming and Testing User's Manual
T	Define a tool **T** or use the selection window to choose one and insert it with the **NC start** key. **Further information:** Programming and Testing User's Manual
3D ROT	The control opens a window for the 3D rotation settings (#8 / #1-01-1). **Further information:** Programming and Testing User's Manual
Active preset	The control opens the preset table in the **Active preset** window. **Further information:** "Preset table *.pr", Page 503
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:** Programming and Testing User's Manual

Button	Meaning
DCM	The control opens the **Dyna. Coll. Monitoring (DCM)** window where you can activate or deactivate Dynamic Collision Monitoring (DCM (#40 / #5-03-1)). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 250
Manual cycles	The machine manufacturer can define manual cycles that you can use by means of this button. The control makes the following manual cycles (#50 / #4-03-1) available: ■ **Calibrate unbalance** Only for the machine manufacturer **Further information:** "Calibrate unbalance (#50 / #4-03-1)", Page 167 ■ **Measure unbalance** Detect the unbalance of current clamping for turning and calculate suggestions for balance weights **Further information:** "Measure unbalance (#50 / #4-03-1)", Page 168
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 165
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 230
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 197
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Tool management ", Page 197

7.2 Moving the machine axes

Application

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

Further information: "Touch probe functions in the Manual operating mode", Page 373

Related topics

- Programming traverse movements
  Further information: Programming and Testing User's Manual
- Executing traverse movements in the **MDI** application
  Further information: "The MDI Application ", Page 369

Description of function

The control offers the following methods for moving axes manually:

- Axis-direction keys
- Incremental jog positioning with the **Jog increment** button
- Traversing with electronic handwheels
  Further information: "Electronic handwheel", Page 519

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

Further information: "Status displays", Page 117

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

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

Further information: "Status overview on the TNC bar", Page 125

7.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:

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

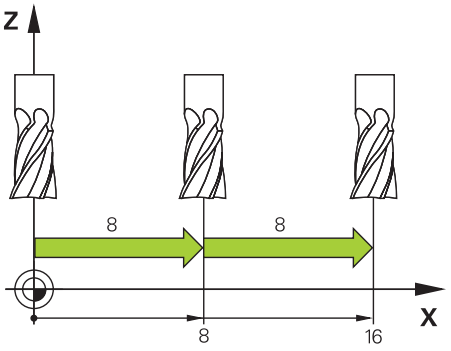
- ▶ Press the axis key of the desired axis
- > The control moves the axis as long as you press the key.

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

You can move more than one axis at a time.

7.2.2 Incremental jog positioning of axes

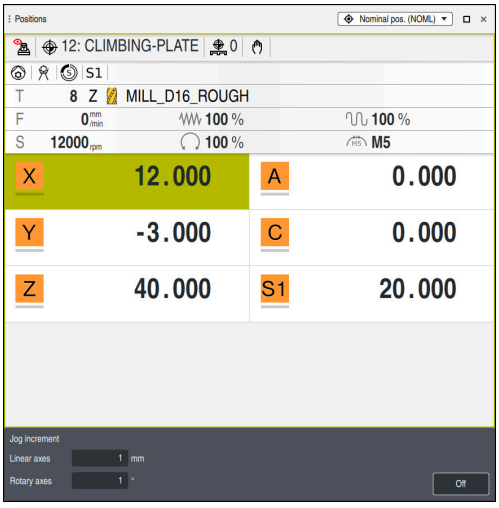
Incremental jog positioning allows you to move a machine axis by a preset distance. The input range for the infeed is from 0.001 mm to 10 mm.



To position an axis incrementally:

- Select the **Manual** operating mode
- Select the **Manual operation** application
- Select **Jog increment**
- > The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
- Enter the jog increment for linear axes and rotary axes
- Press the axis key of the desired axis
- > The control positions the axis in the selected direction by the defined jog increment.
- Select **Off**
- > The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

> You can also end incremental positioning with the **Jog inc. on** button.



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

Note

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

7.3 Unbalance functions (#50 / #4-03-1)

7.3.1 Overview

The control provides the following unbalance functions:

Function	Meaning	Further information
Calibrate unbalance	Specify the unbalance reference values Only for the machine manufacturer	Page 167
Measure unbalance	Detect the unbalance of current clamping for turning and calculate suggestions for balance weights	Page 168

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)

Refer to your machine manual.

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

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

7.3.2 Calibrate unbalance (#50 / #4-03-1)

Application

The unbalance calibration is performed by the machine manufacturer before shipping the machine. With unbalance calibration, the rotary table is operated at various speeds with a defined weight mounted at a defined radial position. The measurement is repeated with different weights.

Related topics

- Determining the unbalance of the current fixture
  Further information: "Measure unbalance (#50 / #4-03-1)", Page 168
- Unbalance fundamentals
  Further information: Programming and Testing User's Manual

Requirements

- Software option Turning (#50 / #4-03-1)
- Function enabled by the machine manufacturer
- **FUNCTION MODE TURN** active

Description of function

NOTICE

Danger of collision!

Changes to the calibration data can lead to undesired behavior. It is not recommended for the machine operator or NC programmer to use the **CALIBRATE UNBALANCE** cycle. There is a risk of collision during the execution of the function and during the subsequent machining!

- Use the function only if agreed upon with the machine manufacturer
- Refer to the machine tool manufacturer's documentation

7.3.3 Measure unbalance (#50 / #4-03-1)

Application

The **MEASURE UNBALANCE** cycle determines the unbalance of the workpiece and calculates the mass and position of a balancing mass.

Related topics

- Cycle **892 CHECK UNBALANCE**
  Further information: User's Manual for Machining Cycles
- Unbalance fundamentals

Requirements

- Software option Turning (#50 / #4-03-1)
- Function enabled by the machine manufacturer
- **FUNCTION MODE TURN** active

Description of function

In the **Unbalance measurement: Speed limitation** window, you define at which speed the control will measure the unbalance.

The control starts rotating the table at a low speed and gradually increases the speed up to the defined value.

After completion of the measurement, the control will display the calculated mass and the radial position of the compensation weight in the **Result diagram** window.

After clamping a balancing weight, the unbalance must be checked again in a measurement.

The Result diagram window

The **Result diagram** window contains the following areas:

Area	Meaning
Determined values	■ **Runout:** Determined unbalance at the defined speed ■ **Shaft speed:** Speed defined in the **Unbalance measurement: Speed limitation** window
Proposed unbalance	Properties and clamping of the ideal compensation weight: ■ **Angle:** Angle on the table ■ **Radial position:** Distance from the table center in mm ■ **Weight [g]:**
Alternative settings	■ **Weight [g]:** ■ **Radial position:** If you wish to use a different radial position or mass for the balancing mass, you can overwrite one value and have the other value recalculated automatically. When you enter a value and press the **RETURN** key, the control will recalculate the value.

The control shows a diagram with the possible mass and radial-position values of the compensation weight. The control marks the **Proposed unbalance** with a circle.

When you have the control recalculate the value, it marks the new value with a red circle.

Note

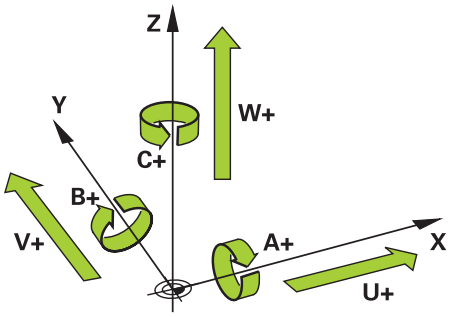
To compensate for an unbalance, several balancing weights at different positions may be required.

8

NC Fundamentals

8.1 NC fundamentals

8.1.1 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.

8.1.2 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:

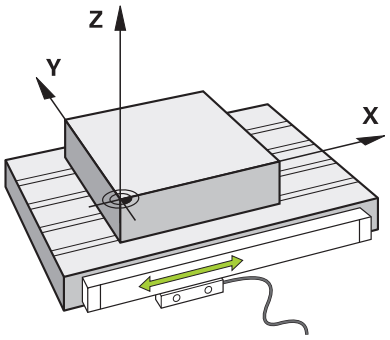
Main axis	Secondary axis	Tool axis	Working plane
X	**Y**	**Z**	**XY**, also **UV**, **XV**, **UY**
Y	**Z**	**X**	**YZ**, also **WU**, **ZU**, **WX**
Z	**X**	**Y**	**ZX**, also **VW**, **YW**, **VZ**

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.

8.1.3 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 214

Position encoders can measure these positions through different methods:

- Absolutely
- Incrementally

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

Absolute position encoders

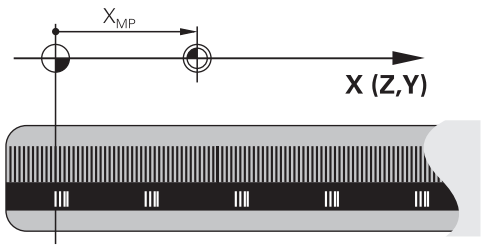
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

Incremental position encoders

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

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

Further information: "Axis reference run", Page 157



8.1.4 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 179

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 216 If you program **M91** in an NC block, the defined values are referenced to the machine datum. **Further information:** Programming and Testing User's Manual
 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:** Programming and Testing User's Manual
	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 173 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 157
	**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 220 The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example. **Further information:** "Preset management", Page 230 **Further information:** Programming and Testing User's Manual If no transformations are defined, the entries in the NC program refer to the workpiece preset.

Icon	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.

9

Tools

9.1 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 179
- Further parameters are needed to completely define the tool. One place to find these parameters is the manufacturer's tool catalog.

  Further information: "Tool parameters", Page 185
- Save all collected parameters of this tool in the tool management.

  Further information: "Tool management ", Page 197
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 201
- After finishing tool definition, program a tool call within an NC program.

  Further information: Programming and Testing User's Manual
- 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: Programming and Testing User's Manual
- If needed, perform a tool usage test before starting the program. This process checks if the tools are available in the machine and have sufficient remaining tool life.

  Further information: "Tool usage test", Page 208
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: Programming and Testing User's Manual

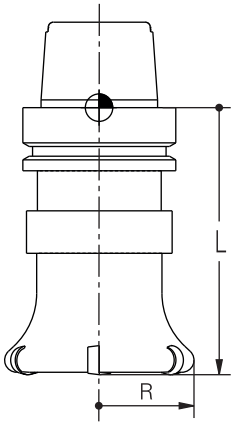
9.2 Presets on the tool

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

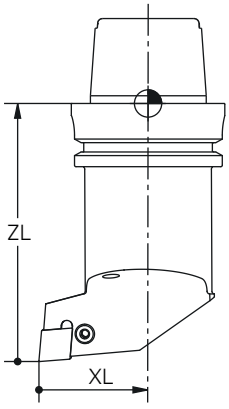
Related topics

- Presets in the machine or on the workpiece
  Further information: "Presets in the machine", Page 174

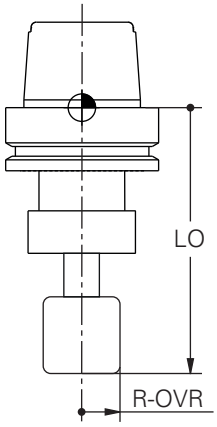
9.2.1 Tool carrier reference point



Milling cutter



Turning tool



Grinding tool

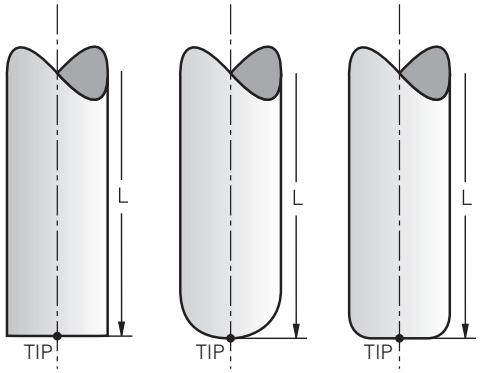
The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

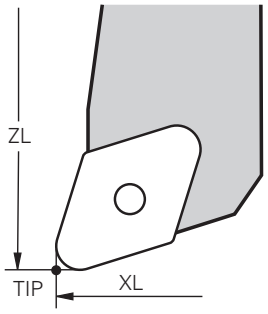
Further information: "Tool management ", Page 197

Further information: "Measuring the tool by scratching", Page 406

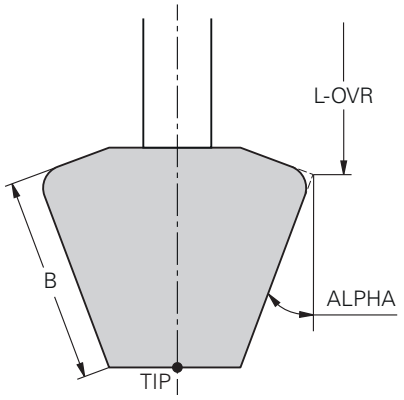
9.2.2 Tool tip TIP



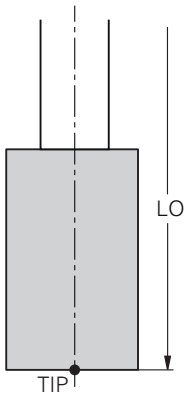
Milling tools



Lathe tools



Grinding tools



Further information: "Tool coordinate system T-CS", Page 227

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: "Tool parameters", Page 185

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.

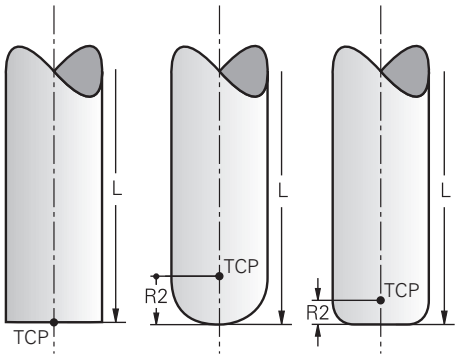
For the following grinding tools, the control calculates the longest point of the tool from several parameters:

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

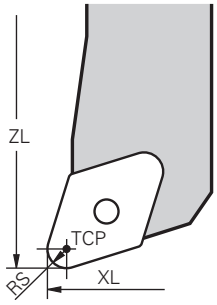
The tool tip is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 182

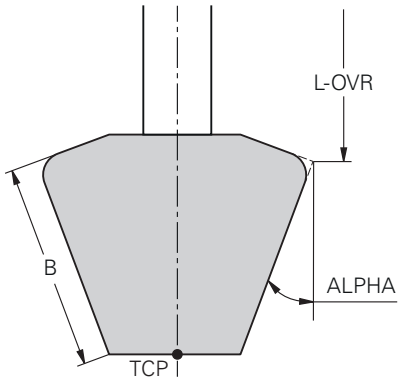
9.2.3 Tool center point (TCP, tool center point)



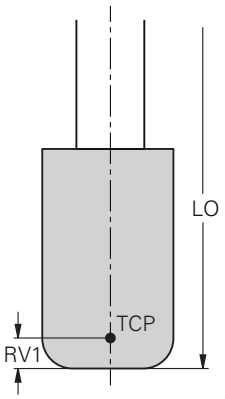
Milling tools



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.

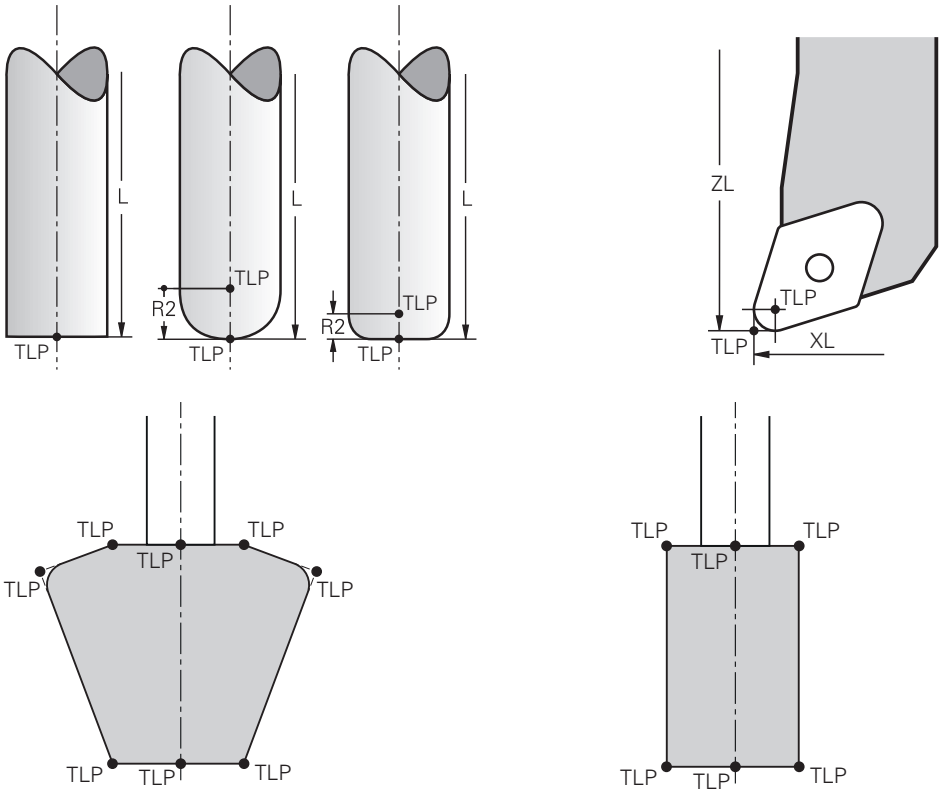
Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool management ", Page 197

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 182

9.2.4 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.

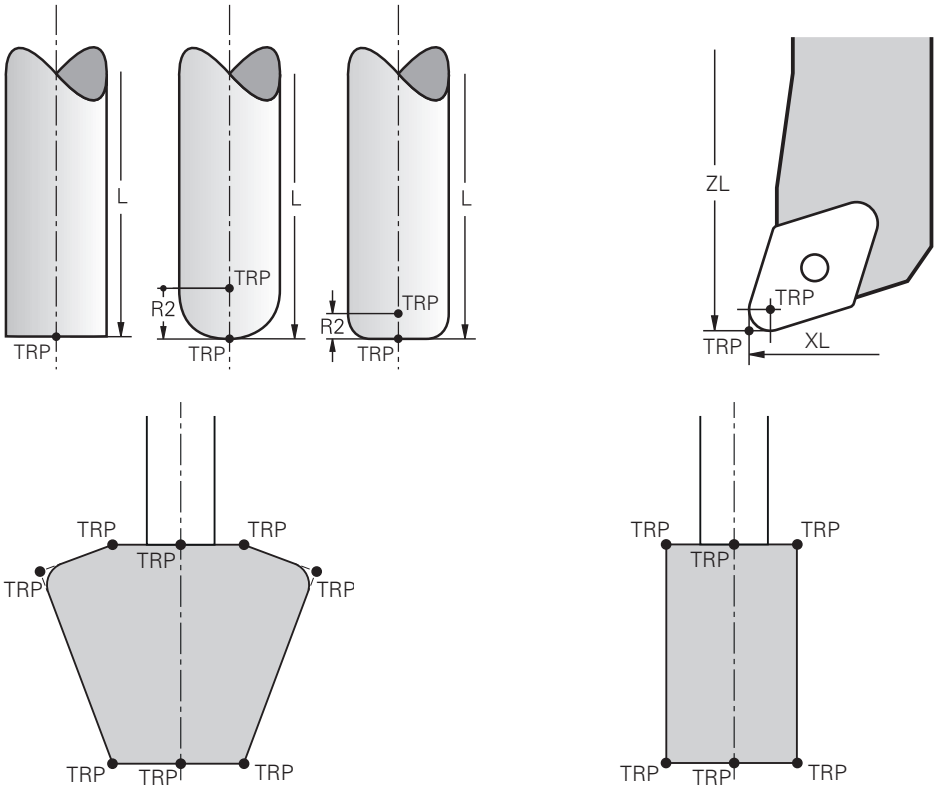
In the function **FUNCTION TCPM** (#9 / #4-01-1), you can also choose the tool location point to be at the tool center point.

Further information: Programming and Testing User's Manual

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: Programming and Testing User's Manual

9.2.5 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.

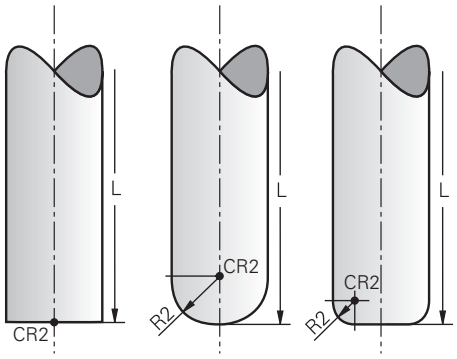
Further information: Programming and Testing User's Manual

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.

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: Programming and Testing User's Manual

9.2.6 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: Programming and Testing User's Manual

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 182

9.3 Tool parameters

Application

In the tool parameters you give the control all the information it need to, for example calculate the contours or run the simulation.

The parameters required depend, for example, on the tool type.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool types
  Further information: "Tool types", Page 194
- Tool tables
  Further information: "Tool tables", Page 457
- Tool table **tool.t**
  Further information: "Parameters of the tool table tool.t", Page 457
- Turning tool table **toolturn.trn** (#50 / #4-03-1)
  Further information: "Parameters of the turning tool table toolturn.trn", Page 468
- Grinding tool table **toolgrind.grd** (#156 / #4-04-1)
  Further information: "Parameters of the grinding tool table toolgrind.grd", Page 476
- Dressing tool table **tooldress.drs** (#156 / #4-04-1)
  Further information: "Parameters of the dressing tool table tooldress.drs", Page 486
- Touch probe table **tchprobe.tp**
  Further information: "Parameters of the touch probe table tchprobe.tp", Page 490

Description of function

There are various possibilities for determining the parameters. For example:

- You can measure your tools in the machine (e. g., with a tool touch probe) or externally with a tool presetter.
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Take further tool information from the manufacturer’s tool catalog (e.g., the material or the number of teeth).

The **Form** workspace in the **Tables** operating mode assists you during parameter input. The control filters the form according the the selected tool type.

HEIDENHAIN recommends entering all known parameters in order to use the following functions to their full extent:

- Simulation
  Further information: Programming and Testing User's Manual
- Machining or touch probe cycles
  Further information: User's Manual for Machining Cycles
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246

9.3.1 Tool ID number

Application

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

Further information: "Tool management ", Page 197

Description of function

The tool ID numbers can be defined in a range from 0 to 32,767.

The tool with the number 0 is defined as the zero tool, with both the length and radius being equal to 0. Upon a TOOL CALL 0, the control unloads the currently used tool and inserts no new tool.

Further information: Programming and Testing User's Manual

9.3.2 Tool name

Application

A tool name can be assigned in addition to the tool ID number. Contrary to the tool ID number, a tool name is not unique.

Description of function

The tool name allows identifying tools easier within the tool management. To this end, key features can be defined such as the diameter or the type of machining (e.g., **MILL_D10_ROUGH**).

As tool names are not unique, assign names that clearly identify the tools.

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z 0 1 2 3 4 5 6 7 8 9 # $ % & , - _ .

When entering lowercase letters, the control replace them with uppercase letters upon saving.

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

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276

Note

- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    Refer to your machine manual.

    If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.

9.3.3 Database ID

Application

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

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

Related topics

- **DB_ID** column of tool management
  Further information: "Tool table tool.t", Page 457

Description of function

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

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

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

Further information: "Indexed tool", Page 188

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

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

9.3.4 Indexed tool

Application

Using an indexed tool, several different parameters can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

Requirement

- Main tool has been defined

Description of function

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

Further information: "Tool carrier reference point", Page 179

Further information: "Creating an indexed tool", Page 189

Examples of an application of indexed tools:

- Step drill

  The parameters of the main tool contain the drill tip, which corresponds to the maximum length. The tool steps are defined are indexed tools. This makes the lengths equal the actual tool dimensions.
- NC center drill

  The main tool is used for defining the theoretical tool tip as the maximum length. This can be used for centering, for example. The indexed tool defines a point along the tool tooth. This can be used for deburring, for example.
- Cut-off milling cutter or T-slot milling cutter

  The main tool is used for defining the lower point of the cutting edge, which equals the maximum length. The indexed tool defines the upper point of the cutting edge. When using the indexed tool for cutting-off, the specified workpiece height can be directly programmed.

Creating an indexed tool

To create an indexed tool:

- Select the **Tables** operating mode
- Select **Tool management**

- Enable **Edit**
- The control enables tool management for editing.

- Select **Insert tool**
- The control opens the **Insert tool** window.
- Select the desired tool type
- Define the tool number of the main tool (e.g., **T5**)

- Press **OK**
- The control adds table row **5**.
- Open the **Form** workspace
- Define all possible parameters in the form, including the maximum tool length

  Further information: "Tool parameters", Page 185

- Select **Insert tool**
- The control opens the **Insert tool** pop-up window.
- Enable the **Index** check box
- The control adds the next free index number for the currently selected tool (e.g., **T5.1**).

- Press **OK**
- The control inserts table row **5.1** with the parameters of the main tool.
- Correct all deviating parameters in the form

  Further information: "Tool parameters", Page 185

The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.

Further information: "Tool carrier reference point", Page 179

Notes

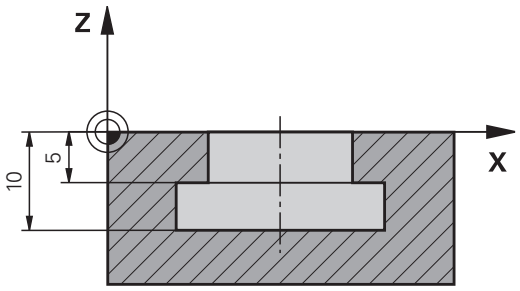
- The control describes some parameters automatically, for example the current tool age **CUR_TIME**. The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 457
- When you create an indexed tool, the control will copy the parameters from the previous table row. The previous table row can either be the main tool or an existing indexed tool.
- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.
- If you delete a main tool, the control will delete all associated indexed tools as well.
- If you copy or cut indexed tools only, you can use **Append** to add the indices to the currently selected tool.

  Further information: Programming and Testing User's Manual
- Up to nine indexed tools can be added to each main tool. Grinding tools do not support indexed tools.
- If you define a replacement tool **RT**, this applies to the respective table row exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This means, for example, that the main tool can still be used.

  Further information: Programming and Testing User's Manual

Example of T-slot milling cutter



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

Two tool definitions are required for producing the T-slot.

- The main tool dimension refers to the lower point of the cutting edge, which equals the maximum tool length. This can be used for machining the bottom edge of the T-slot.
- The dimension of the indexed tool refers to the upper point of the cutting edge. This can be used for machining the top edge of the T-slot.

Please ensure that all required parameters are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table rows.

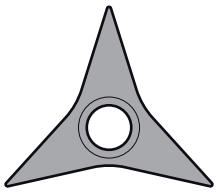
The T-slot is programmed in two machining steps:

- The 10 mm depth is programmed with the main tool.
- The 5 mm depth is programmed with the indexed tool.

11 TOOL CALL 7 Z S2000	; Call the main tool
12 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
13 L Z-10 R0 F500	; Move to machining depth
14 CALL LBL "CONTOUR"	; Machine the bottom edge of the T-slot with the main tool
*** - ...**	
21 TOOL CALL 7.1 Z F2000	; Call the indexed tool
22 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
23 L Z-5 R0 F500	; Move to machining depth
24 CALL LBL "CONTOUR"	; Machine the top edge of the T-slot with the indexed tool

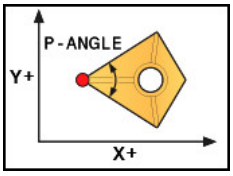
Example: FreeTurn tool (#50 / #4-03-1)

You need the following parameters for a FreeTurn tool:



FreeTurn tool with three finishing teeth

Integrating information about the point angles **P-ANGLE** and the tool length **ZL** (for example, **FT1_35-35-35_100**) into the tool name is recommended.

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	The tool length **ZL** equals the total tool length, relating to the tool carrier preset. **Further information:** "Presets on the tool", Page 179
 **XL**	Tool length 2	The tool length **XL** equals the difference between the spindle center and the tool tip of the tooth. **XL** must always be defined as a negative value with FreeTurn tools. **Further information:** "Presets on the tool", Page 179
 **YL**	Tool length 3	The tool length **YL** is always 0 with FreeTurn tools.
 **RS**	Cutting radius	You can take the radius **RS** from the tool catalog.
 **TYPE**	Lathe tool type	You select between a rough-turning tool (**ROUGH**) and finishing tool (**FINISH**). **Further information:** "Technology-specific tool types", Page 195
 **TO**	Tool orientation	The tool orientation **TO** is always 18 with FreeTurn tools. 
 **ORI**	Angle of orientation	The angle of orientation **ORI** defines the offset of the single teeth with respect to one another. If the first tooth has the value 0, define the second tooth of symmetrical tools at 120 and the third tooth at 240.

Icon and parameter	Meaning	Intended use
 **P-ANGLE**	Point angle	You can get the point angle **P-ANGLE** from the tool catalog.
 **CUTLENGTH**	Usable tooth length	You can get the usable tooth length **CUTLENGTH** from the tool catalog.
 **KINEMATIC**	Tool-carrier kinematics	Using the optional tool-carrier kinematics, the control can, for example, monitor the tool for collisions. Assign the same kinematics to each single tooth.

9.3.5 Tool types

Application

The control uses the tool types to filter the parameters that you can edit in tool management.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool tables
  Further information: "Tool tables", Page 457

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected using the **TYP** parameter of the tool management:

Icon	Tool type	Number
	Milling cutter (**MILL**)	0
	Rough cutter (**MILL_R**)	9
	Finishing cutter (**MILL_F**)	10
	Face mill (**MILL_FACE**)	14
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Chamfer mill (**MILL_CHAMFER**)	24
	Side milling cutter (**MILL_SIDE**)	25
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4
	Turning tool (**TURN**) (#50 / #4-03-1) **Further information:** "Turning tool types (#50 / #4-03-1)", Page 195	29
	Touch probe (**TCHP**) (#17 / #1-05-1)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5
	Piloted counterbore (**TSINK**)	6

Icon	Tool type	Number
	Boring tool (**BOR**)	7
	Back boring tool (**BCKBOR**)	8
	Thread miller (**GF**)	15
	Thread miller with chamfer (**GSF**)	16
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19
	Circular thread mill (**ZBGF**)	20
	Grinding wheel (**GRIND**) (#156 / #4-04-1) **Further information:** "Grinding tool types (#156 / #4-04-1)", Page 196	30
	Dressing tool (**DRESS**) (#156 / #4-04-1) **Further information:** "Dressing tool types (#156 / #4-04-1)", Page 196	31

Technology-specific tool types

Depending on the selected tool type **TYP**, you can use the **TYPE** parameter of the tool management to define a technology-specific tool type and thus specify the tool type more closely. The control offers the **TYPE** parameter for the **TURN**, **GRIND** and **DRESS** tool types.

Turning tool types (#50 / #4-03-1)

Select between the types below within the turning tools:

Icon	Technology-specific tool type	Number
	Rough-turning tool (**ROUGH**)	11
	Finish-turning tool (**FINISH**)	12
	Thread-turning tool (**THREAD**)	14
	Recessing tool (**RECESS**)	15
	Button tool (**BUTTON**)	21
	Recess-turning tool (**RECTURN**)	26

Grinding tool types (#156 / #4-04-1)

Select between the types below within the grinding tools:

Icon	Technology-specific tool type	Number
	Cylindrical grinding pin (**PIN**)	1
	Conical grinding pin (**CONE**)	2
	Cup wheel (**CUP**)	3
	Straight wheel (**CYLINDER**)	26
	Slant wheel (**ANGULAR**)	27
	Facing wheel (**FACE**)	28

Dressing tool types (#156 / #4-04-1)

Select between the types below within the dressing tools:

Icon	Technology-specific tool type	Number
	Stationary dresser with radius (**FIXRADIUS**)	101
	Rotating dresser with radius (**ROTRADIUS**)	103
	Stationary dresser (flat) (**FIXFLAT**)	110
	Rotating (flat) (**ROTFLAT**)	120

9.4 Tool management

Application

The control displays the tool definitions of all technologies as well as the tools currently present in the tool magazine in the **Tool management** application of **Tables** operating mode.

The tool management allows you to add or delete tools, or to edit parameters.

Related topics

- Creating new tools
  Further information: "Setting up a tool", Page 107
- Table workspace
  Further information: "The Table workspace", Page 447
- Form workspace
  Further information: "The Form workspace for tables", Page 454

Description of function

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

The control displays all parameters of the tool tables below in the tool management:

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 457
- Turning-tool table **toolturn.trn** (#50 / #4-03-1)
  Further information: "Turning tool table toolturn.trn (#50 / #4-03-1)", Page 468
- Grinding-tool table **toolgrind.grd** (#156 / #4-04-1)
  Further information: "Grinding tool table toolgrind.grd (#156 / #4-04-1)", Page 473
- Dressing-tool table **tooldress.drs** (#156 / #4-04-1)
  Further information: "Dressing tool table tooldress.drs (#156 / #4-04-1)", Page 485
- Touch-probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 489

The control additionally displays the pockets occupied in the magazine from pocket table **tool_p.tch** in the tool management.

Further information: "Pocket table tool_p.tch", Page 494

You can edit the parameters in the **Table** or **Form** workspaces. In the **Form** workspace the control shows the correct parameters for each tool type.

Further information: "Tool parameters", Page 185

Notes

- When creating a new tool, the length **L** and radius **R** parameters are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.
- Tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.
- When editing parameters, bear in mind that the current tool may have been entered in the **RT** column as a replacement tool of another tool!
- Make sure to keep the tool table as short and clear as possible so that it does not impair the computing speed of your control. Use a maximum of 10,000 tool entries in tool management. For example, you can delete all unused tool numbers; tool numbers need not be sequential.
- If the cursor is within the **Table** workspace and the **Edit** toggle switch is deactivated, a search using the keyboard can be started. The control opens a separate window with an input field and automatically searches for the entered string. If the controls finds a tool with the entered characters, it selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window.
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

  Further information: Programming and Testing User's Manual

9.4.1 Importing and exporting tool data

Application

The control can import and export tool data. This avoids manual editing efforts and possible typing errors. Importing tool data is particularly useful in connection with a tool presetter. Exported tool data can be used for the tool database of your CAM system, for example.

Description of function

The control transmits tool data as a CSV file.

Further information: Programming and Testing User's Manual

The tool data transfer file is structured as follows:

- The first row contains the tool table column names that are transferred.
- The other rows contain the parameters to be transferred. The order of the parameter values must match the order of the column names in the first row. Decimal numbers are separated by a point.

The column names and the tool parameter values are placed in double quotation marks and are separated by semicolons.

Please note the following regarding the transfer file:

- The tool number must be present.
- Any tool data can be imported. The data record does not need to contain all tool table column names or all parameter values.
- Missing parameter values contain no value between the quotation marks.
- The column names can be arranged in any order. The sequence of parameter values must match the order of column names.

Importing tool data

To import tool data:

- Select the **Tables** operating mode
- Select **Tool management**

- Enable **Edit**
- The control enables tool management for editing.

- Select **Import**
- The control opens a selection window.
- Select the desired CSV file

- Select **Import**
- The control adds the tool data to the tool management.
- If required, the control opens the **Confirm import** window (e.g., in case of identical tool numbers).
- Selecting the procedure:
  - **Append**: the control adds the tool data as new rows at the end of the table.
  - **Overwrite**: the control overwrites the initial tool data with the tool data from the transfer file.
  - **Cancel**: the control cancels the import process.

NOTICE

Caution: Data may be lost!

When overwriting existing tool data with the **Overwrite** function, the control will permanently delete the initial tool data!

- Use this function only with tool data that are no longer needed

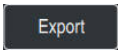
Exporting tool data

To export tool data:

- Select the **Tables** operating mode
- Select **Tool management**

- Enable **Edit**
- The control enables tool management for editing.
- Mark the tool to be exported
- Open the context menu with a long press or by right-clicking

  Further information: Programming and Testing User's Manual
- Select **Mark row**
- Mark further tools if required



- Select **Export**
- The control opens the **Save as** window.
- Select a path

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

- Enter the file name
- Select the file type

> You can export the following CSV formats:
> - **TNC7 (semicolon-separated)**
> - **iTNC 530 / TNC 640 (comma-separated)**

- Select **Create**
- The control will save the file using the selected path.

Notes

NOTICE

Caution: Possible material damage!

If the transfer file contains unknown column names, the control will not accept the data from this column! In this case, the control will perform the operations with an incompletely defined tool.

- Check whether the column names are correct
- After importing, check the tool data and adapt them if required.

- The transfer file must be saved under **TNC:\table**.
- The control creates an output of the CSV files with the following formatting:
  - **TNC7 (semicolon-separated)** encloses the values in double quotation marks, the individual values are separated by semicolons
  - **iTNC 530 / TNC 640 (comma-separated)** encloses the values in double curly brackets, the individual values are separated by commas

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

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

9.5 Tool carrier management

Application

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

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

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

Related topics

- The **Simulation** workspace
  Further information: Programming and Testing User's Manual
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
- Adding a tool model to the tool definition (#140 / #5-03-2)
  Further information: "Tool model (#140 / #5-03-2)", Page 205
- Validating a 3D model for the tool carrier (#56-61 / #3-02-1*)
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590

Requirements

- Kinematics description
  The machine manufacturer creates the kinematics description
- Insertion point defined
  The machine manufacturer defines the insertion point for the tool carrier.
- Tool carrier model exists
  You must save the tool carrier model in the **Toolkinematics** folder.
  Path: **TNC:\system\Toolkinematics**
- The tool carrier model has been assigned to the tool
  Further information: "Assigning a tool carrier", Page 202

Description of function

The tool carrier model must meet the following requirements:

- Use permitted characters for the file name
  Further information: Programming and Testing User's Manual
- Use a supported format
  - CFG file
  - M3D file
  - STL file
    - Max. 20 000 triangles
    - Triangular mesh forms a closed shell
      Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356

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

Further information: "Customizing tool carrier templates with ToolHolderWizard", Page 203

9.5.1 Assigning a tool carrier

To assign a tool carrier to a tool:

- Select the **Tables** operating mode
- Select **Tool management**
- Select the tool you want to use

- Enable **Edit**
- If applicable, open the **Form** workspace
- In the **Additional geometry data** panel, select the **KINEMATIC** parameter
- > The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- Select the desired tool carrier

- Select **OK**
- > The control assigns the 3D model of the tool carrier to the tool.

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

Notes

- Sample files for tool carrier templates are available on the programming station in the **TNC:\system\Toolkinematics** folder.
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

  Further information: Programming and Testing User's Manual
- On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.

  HEIDENHAIN recommends using **Z** as the tool axis for machining. Using the Adv. Function Set 1 (#8 / #1-01-1) software option, you can tilt the machining plane to the angle of the exchangeable angle heads and thus continue working with the tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM (#40 / #5-03-1)). Thus, the tool carriers are protected against collisions with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
- 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.
- When you are dressing a tool, the control hides the tool carrier in the **Simulation** workspace.

  Further information: Programming and Testing User's Manual
- You cannot assign tool-carrier kinematics descriptions with transformations, such as angle heads, to grinding tools.

9.6 Customizing tool carrier templates with ToolHolderWizard

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. HEIDENHAIN provides ready-to-use tool carrier templates for downloading. Tool carrier templates are 3D models with fixed geometries but editable dimensions.

They can be downloaded through the following link:

HEIDENHAIN NC solutions

If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

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

Further information: "Parameterizing tool carrier templates", Page 204

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

Icon	Meaning
	**Close the application**
	**Open file**
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Show or hide **Transformation vectors**
	Show or hide **Names of collision objects**
	Show or hide **Test points**
	Show or hide **Measuring points**
	**Redo** (restore) the initial view
	**Orientations** (e.g,. top view)

9.6.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

- Select the **Files** operating mode

- Open the **TNC:\system\Toolkinematics** folder
- Double-tap or double-click desired tool carrier template with the ***.cft** extension
- The control opens the **ToolHolderWizard** window.
- Define the dimensions in the **Parameter** area
- Define a name with the ***.cfx** extension in the **Output file** area
- Select **Generate file**
- The control shows the message that the tool carrier template was successfully generated and saves the file in the folder **TNC:\system\Toolkinematics**.
- Select **OK**

- Select **Close the application**

Parameterized tool carriers can consist of several subfiles. If the subfiles are incomplete, the control will display an error message.

Only use fully parameterized tool carriers and error-free STL or M3D files!

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

Application

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

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

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

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

Related topics

- The **Simulation** workspace
- Dynamic Collision Monitoring (DCM (#40 / #5-03-1))
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
- Tool carrier management
  Further information: "Tool carrier management", Page 201
- Validating 3D models with **OPC UA NC Server** (#56-61 / #3-02-1*)
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590

Requirements

- Collision Monitoring v2 (#140 / #5-03-2) software option
- The tool has been defined in tool management
  Further information: "Tool management ", Page 197
- A suitable tool model exits
  You must save the tool model in the **Toolshapes** folder.
  Path: **TNC:\system\Toolshapes**
  Further information: "Tool model requirements", Page 206
- The tool model has been assigned to the tool
  Further information: "Assigning a tool model", Page 207

Description of function

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

- Milling tools
- Drilling tools
- Touch probes

Further information: "Tool types", Page 194

Tool model requirements

General requirements

The tool model must meet the following general requirements:

- Use permitted characters for the file name

  Further information: Programming and Testing User's Manual
- Use a supported format
  - M3D file
  - STL file
    - Max. 20 000 triangles
    - Triangular mesh forms a closed shell

      Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356

> For tool models, the same requirements with respect to STL and M3D files apply as for fixtures.
>
> **Further information:** "Options for fixture files", Page 254

Coordinate system requirements

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

- The Z axis is the rotary axis of the tool model.

  The control will align the tool model parallel to the tool coordinate system **T-CS**.

  Further information: "Tool coordinate system T-CS", Page 227
- The coordinate origin of the 3D model must be identical to the measured point of the tool. If you measure the tool at the tool tip, you also need to set the coordinate origin of the 3D model to the tool tip.

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

Further information: "Tool tip TIP ", Page 180

Further information: Programming and Testing User's Manual

9.7.1 Assigning a tool model

To assign a tool model to a tool:

- Select the **Tables** operating mode
- Select **Tool management**
- Select the tool you want to use

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

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

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

Notes

- The control will always take an assigned tool model into account (e.g., for the tool radius **R=0**). The simulation shows the correct shape of the tool model (e.g., in conjunction with a CAM output (center path)).
- When you delete a tool, make sure to remove the tool model from the **Toolshapes** folder as well. This way, you can avoid that the tool model is accidentally referenced for another tool.
- The **LCUTS** column of the tool table is independent of the datum of the tool model. The value is measured from the tool tip of the tool and is effective in the positive Z axis direction.

  Further information: "Tool table tool.t", Page 457
- 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.

9.8 Tool usage test

Application

The tool usage test allows checking the tools used in the NC program before starting the program. The control checks if the tools used are available in the machine magazine and have sufficient remaining tool life. Any missing tools can be stored in the machine or tools can be exchanged due to insufficient remaining tool life before starting the program. This avoids interruptions while the program is running.

Related topics

- Contents of the tool usage file
  Further information: "Tool usage file", Page 497
- Tool usage test in Batch Process Manager (#154 / #2-05-1)
  Further information: Programming and Testing User's Manual

Requirements

- To perform a tool usage test, you need a tool usage file
  In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.
  Further information: "Tool usage file", Page 497
- The **Generate tool-usage file** setting is set to **Once** or **Always**
  Further information: "Channel Settings", Page 562
- Use the same tool table for the simulation as for the program run
  Further information: Programming and Testing User's Manual

Description of function

Creating the tool usage file

A tool usage file must be generated for performing the tool usage test.

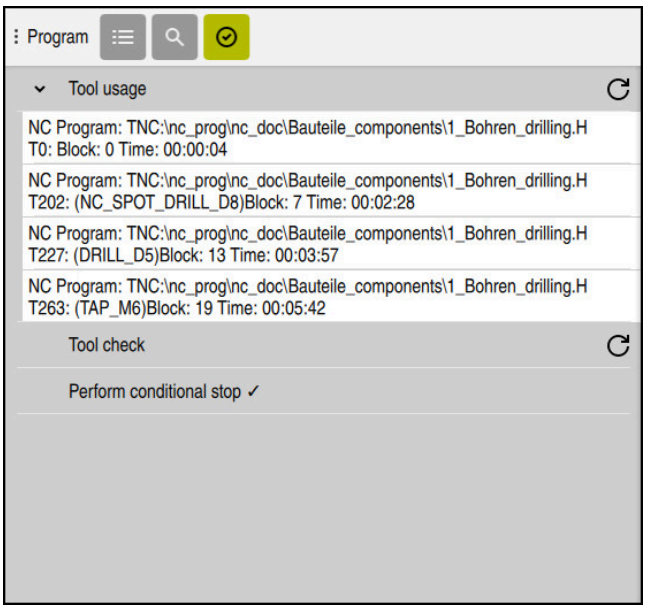
When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

- Simulating the NC program completely
- Executing the NC program completely
- Select the **Refresh** icon in the **Tool usage** area of the **Tool check** column

The control saves the tool usage file with the ***.t.dep** extension in the same folder where the NC program is stored.

Further information: "Tool usage file", Page 497

The Tool check column in the Program workspace



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

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

- **Tool usage**

  Further information: "The Tool usage area", Page 209
- **Tool check**

  Further information: "The Tool check area", Page 210
- **Perform conditional stop**

  Further information: "Override controller", Page 537

Further information: Programming and Testing User's Manual

The Tool usage area

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

Further information: "Creating the tool usage file", Page 208

Further information: "Tool usage file", Page 497

The control displays the chronological order of all tool calls in the **Tool usage** area, along with the following information:

- Path of NC program in which the tool is called
- Tool number and possibly tool name
- Row number of tool call in NC program
- Tool usage time between the tool changes

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

The Tool check area

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

Further information: "Performing the tool usage test", Page 210

When performing the tool usage test, the control checks the following:

- The tool is defined in the tool management

  Further information: "Tool management ", Page 197
- The tool is defined in the pocket table

  Further information: "Pocket table tool_p.tch", Page 494
- The tool has sufficient remaining tool life

  The control checks if the remaining tool life **TIME1** minus **CUR_TIME** is sufficient for the machining process. To meet this requirement, the remaining tool life must be longer than the tool usage time **WTIME** from the tool usage file.

  Further information: "Tool table tool.t", Page 457

  Further information: "Tool usage file", Page 497

The control displays the following information in the **Tool check** area:

- **OK**: All tools are available and have sufficient remaining tool life
- **No suitable tool**: The tool is not defined in the tool management

  In this case, check if the correct tool is selected in the tool call. Otherwise, create the tool in the tool management.
- **External tool**: The tool is defined in the tool management, but not in the pocket table

  If your machine is equipped with a magazine, position the missing tool in the magazine.
- **Insufficient remaining tool life**: The tool is blocked or does not have sufficient remaining tool life

  Change the tool or use a replacement tool.

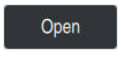
  Further information: Programming and Testing User's Manual

9.8.1 Performing the tool usage test

To perform a tool usage test:

- ▶ Select the **Editor** operating mode

- ▶ Select **Add**
- ▶ Select the desired NC program



- ▶ Select **Open**
- > The control opens the NC program in a new tab.

- ▶ Open the **Tool check** column

- ▶ In the **Tool usage** area, select **Refresh**
- > The control generates a tool usage file and displays the tools used in the **Tool usage** area.

  Further information: "Tool usage file", Page 497

- ▶ In the **Tool check** area, select **Refresh**
- > The control performs the tool usage test.
- > The **Tool check** area shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you double-tap or double-click a tool entry in the **Tool usage** or **Tool check** areas, the control switches to the tool selected in tool management. You can make modifications as needed.
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.

  Further information: Programming and Testing User's Manual
- The control saves the tool usage file as a dependent file (***.dep**).

  Further information: "Tool usage file", Page 497
- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

  Further information: Programming and Testing User's Manual
- The control displays the order of tool calls of the currently running NC program in the **T usage order** (#93 / #2-03-1) table.

  Further information: "T usage order (#93 / #2-03-1)", Page 500
- An overview of all tool calls of the NC program active in the program run is displayed by the control in the **Tooling list** table (#93 / #2-03-1).

  Further information: "Tooling list (#93 / #2-03-1)", Page 502
- The function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- The function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. After **IDX** you define the pallet table row.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPal** (no. 129802) to define whether the control automatically generates a tool usage file upon selecting a pallet table.

10

Coordinate transformation

10.1 Reference systems

10.1.1 Overview

A control requires unambiguous coordinates in order to move an axis to a defined position correctly. For coordinates to be unambiguous, they not only require the values but also a reference system in which these values are valid.

The control differentiates between the following reference systems:

Abbreviation	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 216
B-CS	Basic coordinate system basic coordinate system	Page 219
W-CS	Workpiece coordinate system workpiece coordinate system	Page 220
WPL-CS	Working plane coordinate system working plane coordinate system	Page 222
I-CS	Input coordinate system input coordinate system	Page 225
T-CS	Tool coordinate system tool coordinate system	Page 227

The control uses different reference systems for different purposes. For example, this makes it possible to always exchange tools at the exact same position while maintaining the possibility of adapting an NC program to the workpiece position.

The reference systems build upon each other. The machine coordinate system **M-CS** is the fundamental reference system. The position and orientation of the following reference systems are determined by transformations of the M-CS.

Definition

Transformations

Translatory transformations each enable a shift along a number line. Rotatory transformations enable a rotation around a point.

10.1.2 Basics of coordinate systems

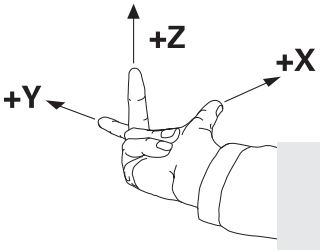
Types of coordinate systems

For coordinates to be unambiguous they must define one point in all axes of the coordinate system:

Axes	Function
One	In a one-dimensional coordinate system, one coordinate defines one point on a number line. Example: on a machine tool, a linear encoder represents a number line.
Two	In a two-dimensional coordinate system, two coordinates define one point in a plane.
Three	In a three-dimensional coordinate system, three coordinates define one point in space.

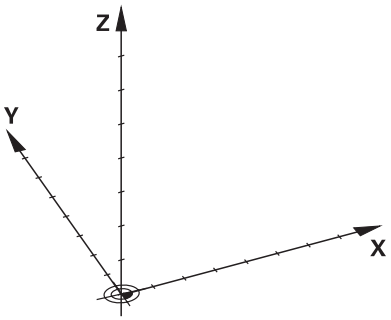
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

Using the right-hand rule you can recreate a three-dimensional Cartesian coordinate system. The fingertips point in the positive directions of the three axes.



Origin of the coordinate system

Unambiguous coordinates require a defined reference point to which the values refer, starting from zero. This point is the coordinate origin, which lies at the intersection of the axes for all three-dimensional Cartesian coordinate systems of the control. The coordinate origin has the coordinates **X+0**, **Y+0**, and **Z+0**.



10.1.3 Machine coordinate system M-CS

Application

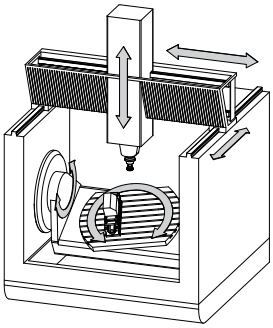
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

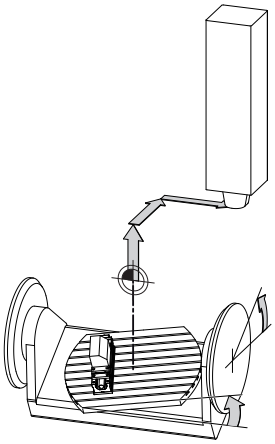
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate system. The **M-CS** thus consists of multiple one-dimensional coordinate systems that correspond to the axes of the machine.

The machine manufacturer defines the position and orientation of the one-dimensional coordinate systems in the kinematics description.



The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

The values in the machine configuration define the zero positions of the position encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can also be located outside of the traverse range.



Position of the machine datum in the machine

Transformations in the machine coordinate system M-CS

The following transformations can be defined in the **M-CS** machine coordinate system:

- Axis-specific shifts in the **OFFS** columns of the preset table

  Further information: "Preset table *.pr", Page 503

  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: Programming and Testing User's Manual

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

  Further information: Programming and Testing User's Manual

- **Additive offset (M-CS)** function for rotary axes in the **GPS** (#44 / #1-06-1) workspace

  Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287

The machine manufacturer can also define further transformations.

Further information: "Note", Page 218

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: Programming and Testing User's Manual

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

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets
- Change pallet presets only after discussion with the machine manufacturer
- Check the pallet preset in the **Setup** application before you start machining

Example

This example illustrates the difference between traverse movements with and without **M91**. The example shows the behavior with a Y axis as oblique axis that is not arranged perpendicularly to the ZX plane.

Traverse movement without M91

```
11 L IY+10
```

You use the Cartesian input coordinate system **I-CS** for programming. The **ACTL.** and **NOML.** modes of the position display show only a movement of the Y axis in the **I-CS**.

The control uses the defined values to determine the required traverse paths of the machine axes. Since the machine axes are not arranged perpendicularly to each other, the control moves the axes **Y** and **Z**.

Since the machine coordinate system **M-CS** is a projection of the machine axes, the **RFACTL** and **RFNOML** modes of the position display show movements of the Y axis and Z axis in the **M-CS**.

Traverse movement with M91

```
11 L IY+10 M91
```

The control moves the machine axis **Y** by 10 mm. The **RFACTL** and **RFNOML** modes of the position display show only a movement of the Y axis in the **M-CS**.

In contrast to the **M-CS**, the **I-CS** is a Cartesian coordinate system; the axes of the two reference systems do not coincide. The **ACTL.** and **NOML.** modes of the position display show movements of the Y axis and Z axis in the **I-CS**.

10.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position and orientation of the workpiece. You determine these values by using a 3D touch probe, for example. The control saves the values in the preset table.

Description of function

Properties of the basic coordinate system B-CS

The basic coordinate system **B-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the end of the kinematics description.

The machine manufacturer defines the coordinate origin and orientation of the **B-CS**.

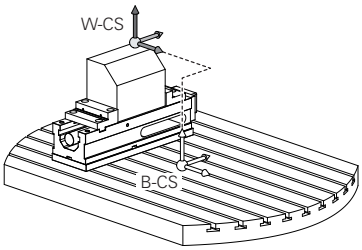
Transformations in the basic coordinate system B-CS

The following columns of the preset table have an effect in the basic coordinate system **B-CS**:

- **X**
- **Y**
- **Z**
- **SPA**
- **SPB**
- **SPC**

You determine the position and orientation of the workpiece coordinate system **W-CS** by using a 3D touch probe, for example. The control saves the determined values as basic transformations in the **B-CS** in the preset table.

Further information: "Preset management", Page 230



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

Further information: "Note", Page 220

Note

The machine manufacturer can define additional basic transformations in the pallet preset table.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets
- Change pallet presets only after discussion with the machine manufacturer
- Check the pallet preset in the **Setup** application before you start machining

10.1.5 Workpiece coordinate system W-CS

Application

In the workpiece coordinate system **W-CS** you define the position and orientation of the working plane. You do this by programming transformations and tilting the working plane.

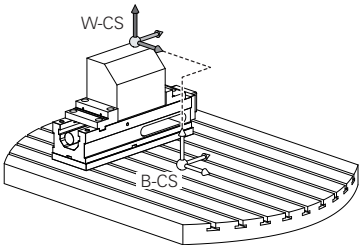
Description of function

Properties of the workpiece coordinate system W-CS

The workpiece coordinate system **W-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the active workpiece preset from the preset table.

Both the position and orientation of the **W-CS** are defined by basic transformations in the preset table.

Further information: "Preset management", Page 230



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: Programming and Testing User's Manual
- Columns **X**, **Y**, **Z** of the datum table before tilting the working plane
  Further information: Programming and Testing User's Manual
- The **TRANS MIRROR** function or Cycle **8 MIRRORING** before tilting the working plane with spatial angles
  Further information: Programming and Testing User's Manual

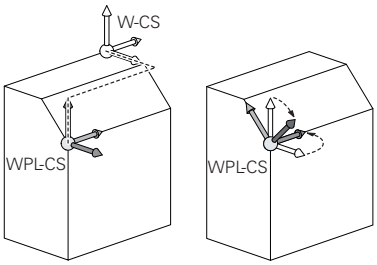
Further information: User's Manual for Machining Cycles

- **PLANE** functions for tilting the working plane (#8 / #1-01-1)

  Further information: Programming and Testing User's Manual

> 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 237

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: "Global Program Settings GPS (#44 / #1-06-1)", Page 287

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 225
- 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 222
- The result of transformations built upon each other depends on the programming sequence.

10.1.6 Working plane coordinate system WPL-CS

Application

In the working plane coordinate system **WPL-CS** you define the position and orientation of the input coordinate system **I-CS** and therefore the reference for the coordinate system in the NC program. You do this by programming transformations after having tilted the working plane.

Further information: "Input coordinate system I-CS", Page 225

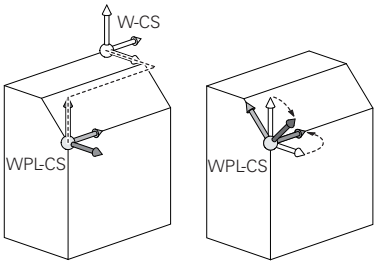
Description of function

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 220

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

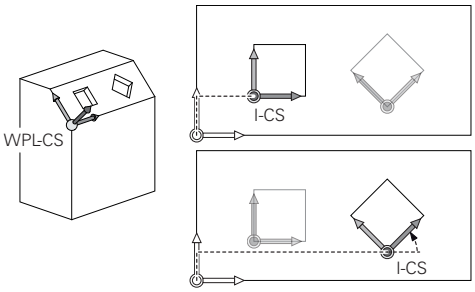


Transformations in the working plane coordinate system WPL-CS

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: Programming and Testing User's Manual
- The **TRANS MIRROR** function or Cycle **8 MIRRORING**
  Further information: Programming and Testing User's Manual
  Further information: User's Manual for Machining Cycles
- The **TRANS ROTATION** function or cycle **10 ROTATION**
  Further information: Programming and Testing User's Manual
  Further information: User's Manual for Machining Cycles
- The **TRANS SCALE** function or cycle **11 SCALING FACTOR**
  Further information: Programming and Testing User's Manual
  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: Programming and Testing User's Manual

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: "Global Program Settings GPS (#44 / #1-06-1)", Page 287

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

> 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:** "The POS tab", Page 137

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**

> 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 225
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
- The result of transformations built upon each other depends on the programming sequence.
- As a **PLANE** function (#8 / #1-01-1), **PLANE RELATIV** is in effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

10.1.7 Input coordinate system I-CS

Application

The programmed values in the NC program refer to the input coordinate system **I-CS**. You use positioning blocks to program the position of the tool.

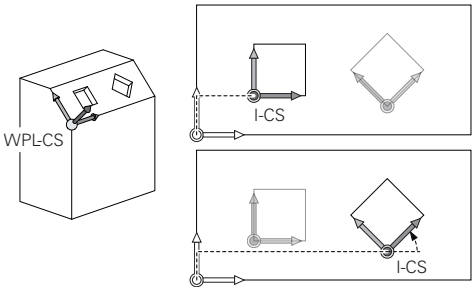
Description of function

Properties of the input coordinate system I-CS

The input coordinate system **I-CS** is a three-dimensional Cartesian coordinate system. You use transformations in the working plane coordinate system **WPL-CS** to define the coordinate origin of the **I-CS**.

Further information: "Working plane coordinate system WPL-CS", Page 222

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 227

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 222

10.1.8 Tool coordinate system T-CS

Application

In the tool coordinate system **T-CS** the control implements tool compensations and tool inclinations.

Description of function

Properties of the tool coordinate system T-CS

The tool coordinate system **T-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the tool tip TIP.

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 174

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 179

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 225

You can use miscellaneous functions to also program in other reference systems, such as **M91** for the machine coordinate system **M-CS**.

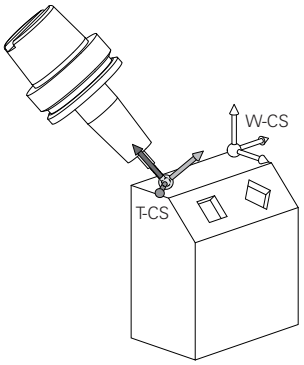
Further information: Programming and Testing User's Manual

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

  Further information: Programming and Testing User's Manual



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: Programming and Testing User's Manual

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: Programming and Testing User's Manual
- Compensation values from the tool call
  Further information: Programming and Testing User's Manual
- Values of the compensation tables ***.tco**
  Further information: Programming and Testing User's Manual
- Values of **FUNCTION TURNDATA CORR T-CS** (#50 / #4-03-1)
  Further information: Programming and Testing User's Manual
- 3D tool compensation with surface normal vectors (#9 / #4-01-1)
  Further information: Programming and Testing User's Manual
- 3D tool radius compensation depending on the contact angle with compensation tables (#92 / #2-02-1)
  Further information: Programming and Testing User's Manual

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: "Global Program Settings GPS (#44 / #1-06-1)", Page 287

The HR 520 and HR 550 FS handwheels show the values of **VT** in the display.

Further information: "Contents of display", Page 526

10.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include, for example, the position and misalignment of a workpiece in the preset table. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 174

Use the preset management in the following cases:

- To tilt the working plane of a machine with table or head rotation axes (#8 / #1-01-1)
- To work on a machine with a head change system
- To machine several workpieces that are clamped at different misaligned positions
- If REF-based datum tables were used on previous control models

Related topics

- Contents of preset table, write protection
  Further information: "Preset table *.pr", Page 503

Description of function

Setting presets

Presets can be set in the following ways:

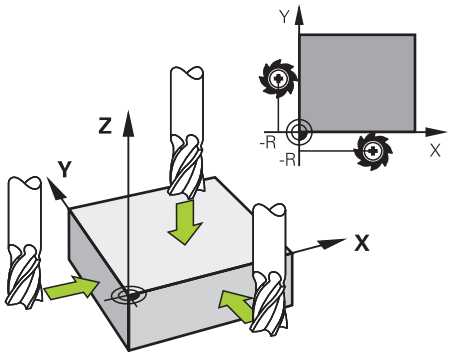
- Setting axis positions manually
  Further information: "Setting a preset manually", Page 233
- Touch probe cycles in the **Setup** application
  Further information: "Touch probe functions in the Manual operating mode", Page 373
- Touch probe cycles in the NC program
  Further information: Measuring Cycles for Workpieces and Tools User's Manual

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 509

Setting a preset with milling cutters

If no workpiece touch probe is available, the preset can also be set by using a milling cutter. In this case, the values are not obtained by probing, but by scratching.



When scratching with a milling cutter, the tool is slowly moved to the workpiece edge in the **Manual operation** application while the spindle is rotating.

As soon as the tool produces chips on the workpiece, the preset is manually set in the desired axis.

Further information: "Setting a preset manually", Page 233

Activating presets

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode
  Further information: "Activating a preset manually", Page 234
- Cycle **247 PRESETTING**
  Further information: User's Manual for Machining Cycles
- **PRESET SELECT** function
  Further information: Programming and Testing User's Manual

When activating a preset, the control resets the following transformations:

- Datum shift with the **TRANS DATUM** function
- Mirroring with the **TRANS MIRROR** function or Cycle **8 MIRRORING**
- Rotation with the **TRANS ROTATION** function or Cycle **10 ROTATION**
- Scaling with the **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
- Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**

Tilting the working plane by using **PLANE** functions or Cycle **19 WORKING PLANE** will not be reset by the control.

Basic rotation and 3D basic rotation

The **SPA**, **SPB** and **SPC** columns define a spatial angle for orienting the workpiece coordinate system **W-CS**. This spatial angle defines the basic rotation or 3D basic rotation of the preset.

Further information: "Workpiece coordinate system W-CS", Page 220

When a rotation around the tool axis is defined, the preset contains a basic rotation (e.g., **SPC** for tool axis **Z**). If one of the remaining columns is defined, the preset contains a 3D basic rotation. If the workpiece preset contains a basic rotation or 3D basic rotation, the control takes these values into account when executing an NC program.

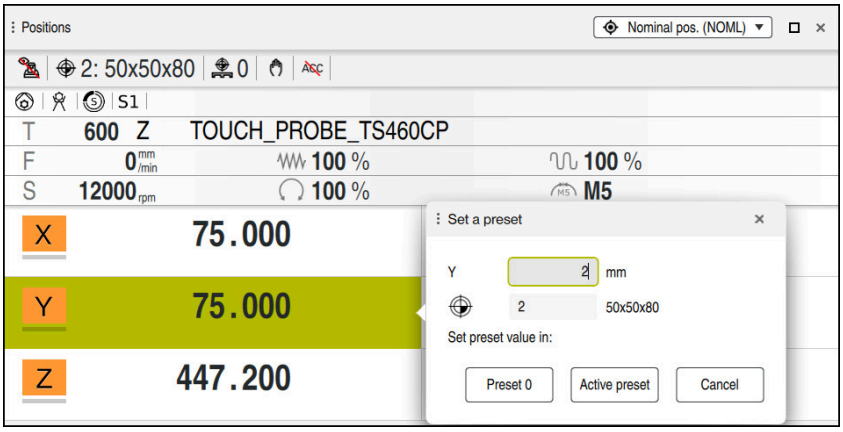
You can use the **3D ROT** (#8 / #1-01-1) button to define whether the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application.

Further information: Programming and Testing User's Manual

When a basic rotation or 3D basic rotation is active, the control displays a symbol in the **Positions** workspace.

Further information: Programming and Testing User's Manual

10.2.1 Setting a preset manually



The **Set a preset** window in the **Positions** workspace

When setting the preset manually, the values can be written either in row 0 of the preset table or in the active row.

To set a preset manually in an axis:

- Select the **Manual operation** application in the **Manual** operating mode
- Open the **Positions** workspace
- Traverse the tool to the desired position (e.g., for scratching)
- Select the row of the desired axis

> The control opens the **Set a preset** window.

- Enter the value of the current axis position, relating to the new preset (e.g., **0**)

> The control activates the **Preset 0** and **Active preset** buttons for selection.

- Select an option (e.g., **Active preset**)

> The control saves the value in the selected preset table row and closes the **Set a preset** window.

> The control updates the values in the **Positions** workspace.

- The **Set the preset** button in the function bar opens the **Set a preset** window for the row marked in green.
- When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

10.2.2 Activating a preset manually

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

To activate a preset manually:

- Select the **Tables** operating mode
- Select the **Presets** application
- Select the desired row

- Select **Activate the preset**

> The control activates the preset.

> The control displays the number and comment of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 119

Further information: "Status overview on the TNC bar", Page 125

Notes

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.
  Further information: Programming and Testing User's Manual

  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.
- Even if the current preset contains a basic rotation or a 3D basic rotation, the **PLANE RESET** function will position the rotary axes at 0° in the **MDI** application.
  Further information: "The MDI Application ", Page 369
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.
  Further information: Programming and Testing User's Manual

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

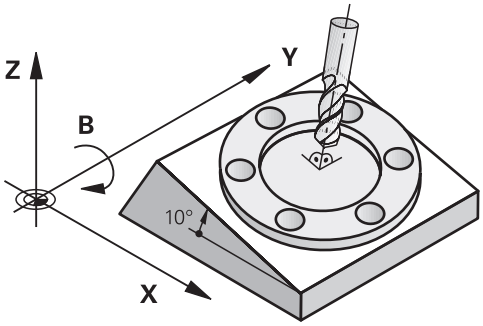
10.3.1 Fundamentals

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 222



Two functions are available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application
  Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 239
- Tilting under program control with the **PLANE** functions in the NC program
  Further information: Programming and Testing User's Manual

> 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 219

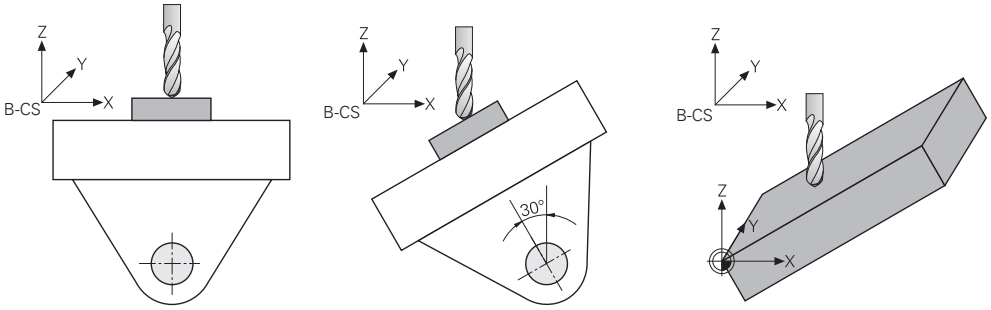
When tilting the working plane, the control moves the machine axes according to the kinematics.

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**.

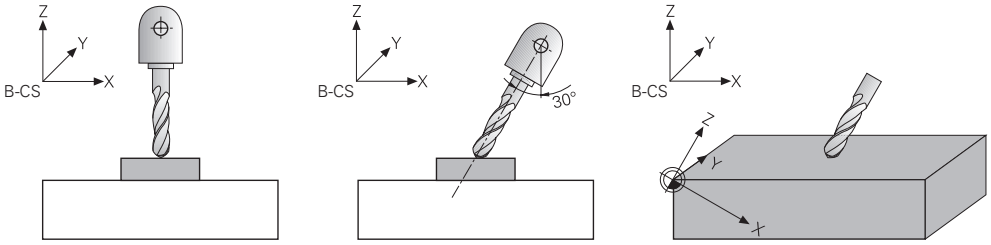
  Further information: "Working plane coordinate system WPL-CS", Page 222



- 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 222



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.

> 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:

- All **PLANE** functions except **PLANE AXIAL**
- Straight line **LN** with vectors (#9 / #4-01-1)
- **FUNCTION TCPM** with **AXIS SPAT** (#9 / #4-01-1) selected

Further information: Programming and Testing User's Manual

> 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: "Comparison of offset and 3D basic rotation", Page 409

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:

- **PLANE AXIAL**
- M function **M128** (#9 / #4-01-1)
- **FUNCTION TCPM** with **AXIS POS** (#9 / #4-01-1) selected

Further information: Programming and Testing User's Manual

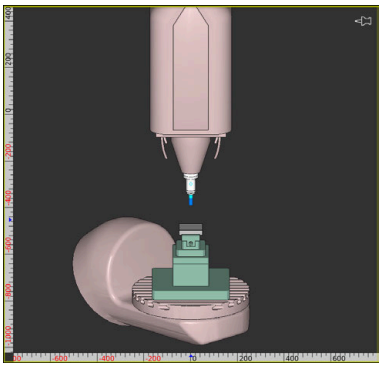
> Even if **M128** or **FUNCTION TCPM** is active with **AXIS POS**, the control will use spatial angles (#9 / #4-01-1) anyway when calculating straight lines **LN** with vectors.

Example: Spatial angles vs. axis angles

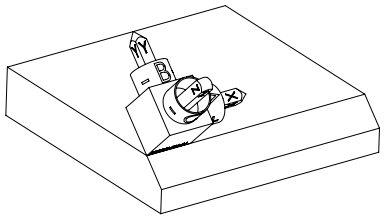
The following example illustrates the difference between spatial angles and axis angles for tilted machining.

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**.



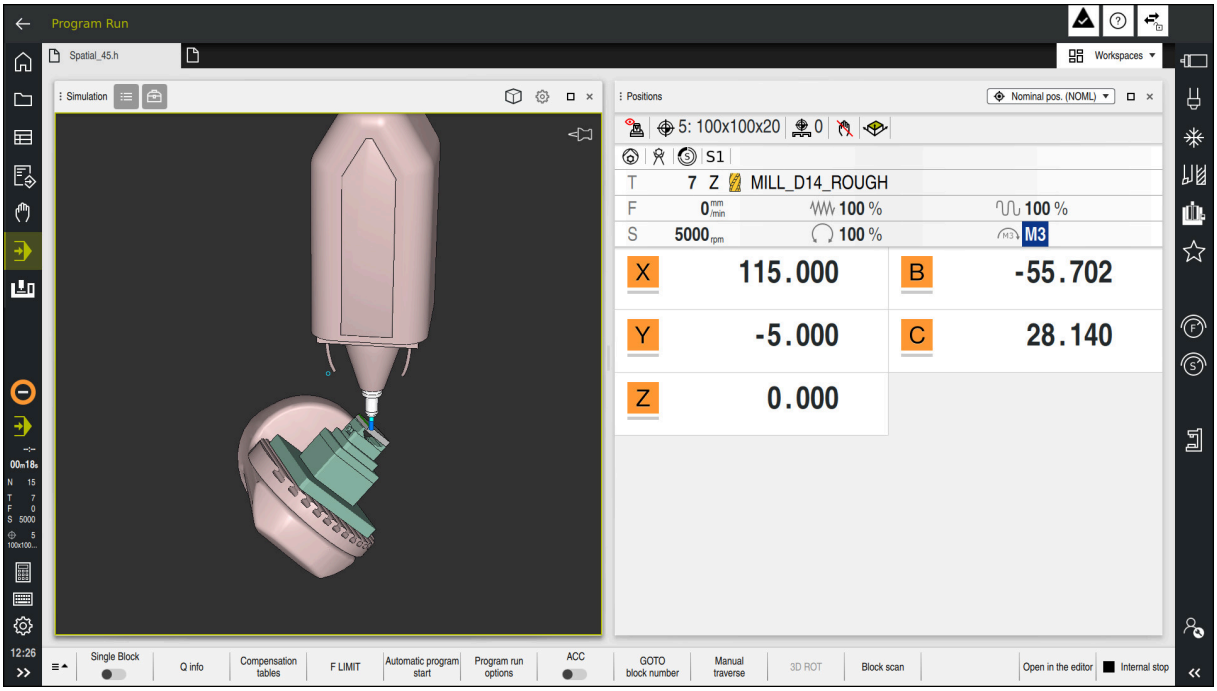
Machine kinematics (not tilted)



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.

10.3.2 The 3-D rotation window (#8 / #1-01-1)

Application

The **3-D rotation** window allows activating and deactivating tilting of the working plane for the **Manual** and **Program Run** operating modes. This allows restoring the tilted working plane and retracting the tool (e.g., after program cancellation in the **Manual operation** application).

Related topics

- Tilting the working plane in the NC program
  Further information: Programming and Testing User's Manual
- Reference systems of the control
  Further information: "Reference systems", Page 214

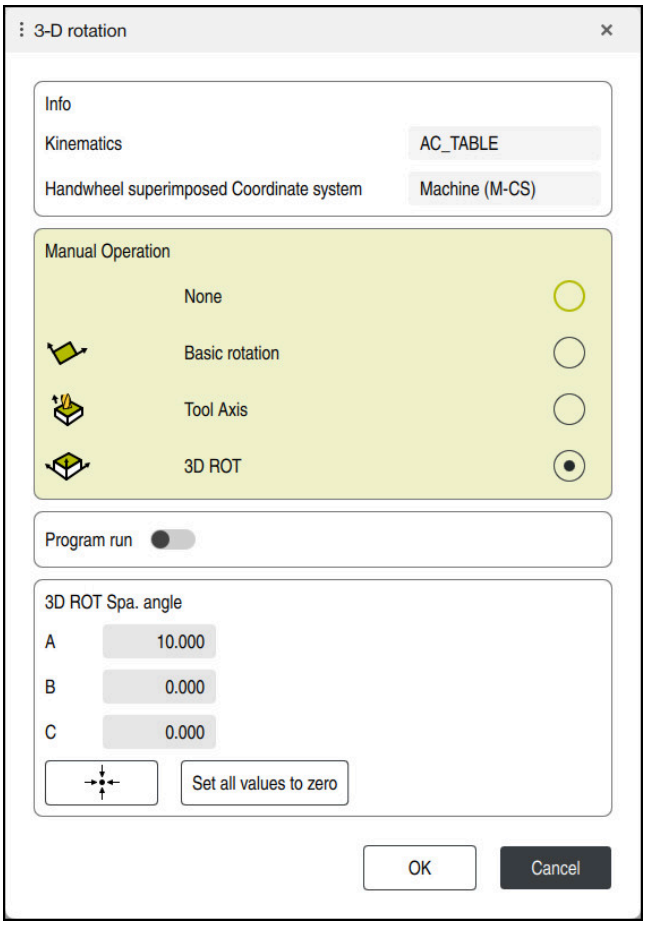
Requirements

- Machine with rotary axes
- Kinematics description
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Software option Adv. Function Set 1 (#8 / #1-01-1)
- Function enabled by the machine manufacturer
  In the machine parameter **rotateWorkPlane** (no. 201201), the machine manufacturer defines whether tilting the working plane is allowed on the machine.
- Tool with tool axis **Z**

Description of function

The **3-D rotation** window can be opened with the **3D ROT** button in the **Manual operation** application.

Further information: "The Manual operation application", Page 162



The **3-D rotation** window

The **3-D rotation** window contains the following information:

Area	Contents
Info	Information about the machine: ■ Name of the active machine kinematics ■ Coordinate system in which handwheel superimpositioning is active **Further information:** "Reference systems", Page 214 **Further information:** "The Handwheel superimp. function", Page 296 **Further information:** Programming and Testing User's Manual

Area	Contents
Manual Operation	Effect of the tilting function in the **Manual** operating mode: ■ **None** The control will ignore rotary axis positions not equal to 0 as well as columns **SPA**, **SPB**, and **SPC** of the preset table. Traverses are effective in the **W-CS** workpiece coordinate system. **Further information:** "Workpiece coordinate system W-CS", Page 220 ■ **Basic rotation** The control takes the columns **SPA**, **SPB** and **SPC** into account, but no rotary axis positions that are not equal to 0. Traverses take place in the **W-CS** workpiece coordinate system. **Further information:** "The Basic rotation selection item", Page 242 ■ **Tool axis** This is relevant only for head rotary axes. The traverses take place in the **T-CS** tool coordinate system. **Further information:** "The Tool axis selection item", Page 242 ■ **3D ROT** The control will consider the positions of the rotary axes and the columns **SPA**, **SPB**, and **SPC** of the preset table. The control moves the axes in accordance with the current tilting situation in the working plane coordinate system **WPL-CS**. **Further information:** "The 3D ROT selection item", Page 242
Program run	Tilting function active or inactive in the **Program Run** operating mode and in the **MDI** application When activating the **Tilt working plane** function for the **Program Run** operating mode, the entered angle of rotation applies starting from the first NC block of the NC program to be run. If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, then the angular values defined there become active. The control will reset the entered angular values to 0.
3D ROT Spa. angle	Currently active angle for the **3D ROT** selection item The machine manufacturer uses the machine parameter **planeOrientation** (no. 201202) to define whether the control calculates with spatial angles **SPA**, **SPB** and **SPC** or with the axis values of the existing rotary axes.

Confirm the selection with **OK**. If a selection item is active in the **Manual Operation** or **Program run** areas, then the control highlights the area in green.

If a selection item is active in the **3-D rotation** window, then the control displays the appropriate symbol in the **Positions** workspace.

Further information: "The Positions workspace", Page 119

The Basic rotation selection item

If you select **Basic rotation**, then the axes move, taking into account a basic rotation or a 3D basic rotation.

Further information: "Basic rotation and 3D basic rotation", Page 232

The axis movements take effect in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 220

If the active workpiece preset contains a basic rotation or 3D basic rotation, the control additionally displays the corresponding icon in the **Positions** workspace.

Further information: "The Positions workspace", Page 119

With this selection item, the **3D ROT Spa. angle** area has no function.

The Tool axis selection item

If you select **Tool axis**, then you can move in the positive or negative direction of the tool axis. The control locks all other axes. This selection item makes sense only for machines with rotary head axes.

The traverse movement is active in the **T-CS** tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 227

This selection item can be used, for example, in the following cases:

- When retracting the tool in the direction of the tool axis during an interruption of a 5-axis machining program.
- When traversing with the axis keys or the handwheel with a pre-positioned tool.

With this selection item, the **3D ROT Spa. angle** area has no function.

The 3D ROT selection item

If you select **3D ROT**, then all axes move in the tilted machining plane. The traversing movements are active in the **WPL-CS** working plane coordinate system.

You can use this selection item, for example, for manual positioning in a program interruption during tilted machining.

Further information: "Working plane coordinate system WPL-CS", Page 222

If a basic rotation or 3D basic rotation has additionally been saved to the preset table, then it will automatically be taken into account.

In the **3D ROT Spa. angle** area, the control shows the currently active angle. The spatial angle can also be edited.

If you edit the values in the **3D ROT Spa. angle** area, then you must position the rotary axes (e.g., in the **MDI** application).

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **COORD ROT**
  - after **PLANE RESET**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer

  COORD ROT is only possible with a free rotary axis.
  Further information: Programming and Testing User's Manual
- The control uses the **TABLE ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **TABLE ROT**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- A tilted working plane will remain active even after a control restart.

  Further information: "The Referencing workspace", Page 157
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

11

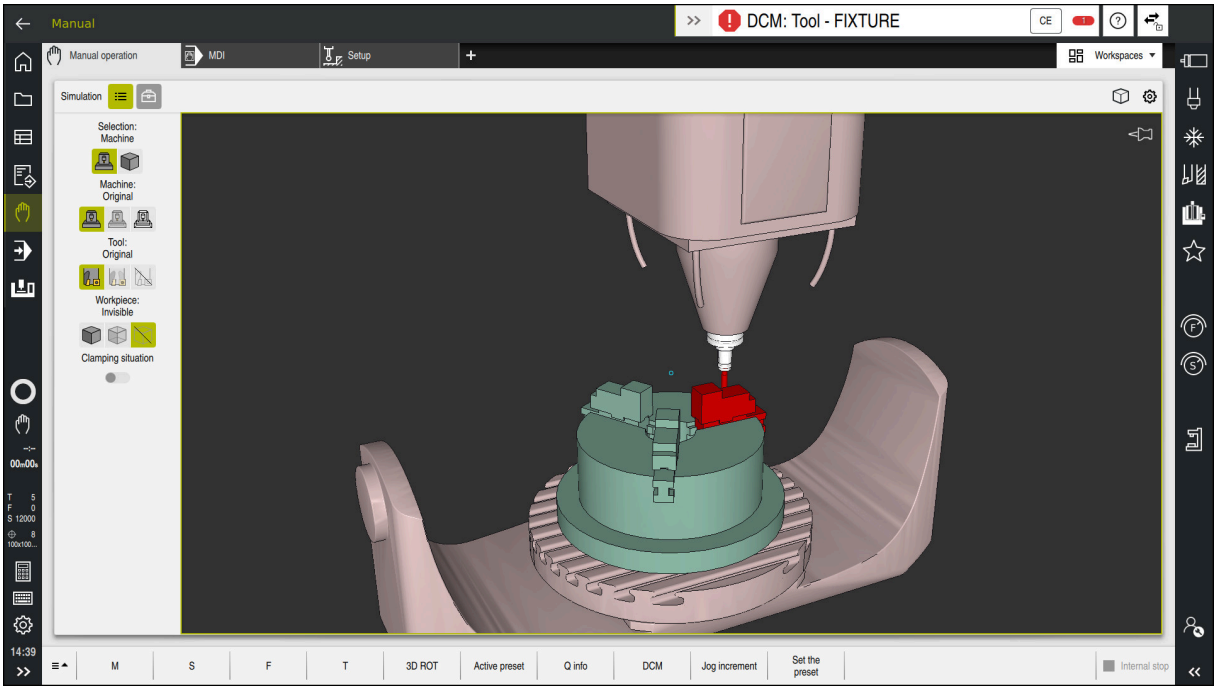
Collision monitoring

11.1 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 253
- Extended tests in the simulation
  Further information: Programming and Testing User's Manual
- Fundamentals of tool carrier management
  Further information: "Tool carrier management", Page 201
- Reduce the minimum clearance between two collision objects (#140 / #5-03-2)
  Further information: Programming and Testing User's Manual

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 253
- Tools with a positive radius **R** and length **L**.

  Further information: "Tool table tool.t", Page 457
- The values in the tool management equal the actual tool dimensions

  Further information: "Tool management ", Page 197

Description of function

Refer to your machine manual.

The machine manufacturer adapts the Dynamic Collision Monitoring (DCM) function to the control.

The machine manufacturer can define machine components and minimum distances to be monitored by the control during all machine movements. If two collision objects come closer to each other than a defined minimum distance, the control generates an error message and terminates the movement.

Error message for Dynamic Collision Monitoring (DCM)

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in **Single Block** mode while DCM is deactivated

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

NOTICE

Danger of collision!

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

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

Further information: Programming and Testing User's Manual

Dynamic Collision Monitoring (DCM) in the Manual and Program Run operating modes

Dynamic Collision Monitoring (DCM) is activated separately for the **Manual** and **Program Run** operating modes, using the **DCM** button.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 250

In the **Manual** and **Program Run** operating modes, the control stops the movement if two collision objects approach each other by less than a minimum clearance. In this case, the control displays an error message naming the two objects causing collision.

Refer to your machine manual.

The machine manufacturer can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of movements. This ensures that the axes stop in good time before a collision occurs.

When the collision warning is triggered, the control displays the colliding objects in red in the **Simulation** workspace.

When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects.

With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.

Dynamic Collision Monitoring (DCM) in the Editor operating mode

Dynamic Collision Monitoring (DCM) is activated for simulation in the **Simulation** workspace.

Further information: Programming and Testing User's Manual

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.

If the **Advanced checks** function is activated, the control will display collisions during simulation (e.g., between workpiece and tool).

Further information: Programming and Testing User's Manual

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: Programming and Testing User's Manual

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 Manual and Program Run operating modes

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in **Single Block** mode while DCM is deactivated

To Dynamic Collision Monitoring (DCM) for the **Manual** and **Program Run** operating modes:

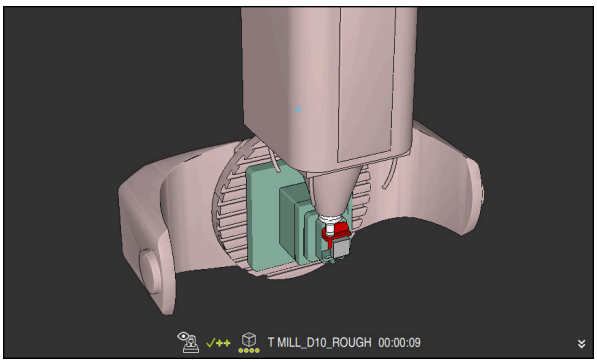
- Select the **Manual** operating mode
- Select the **Manual** application
- Select **DCM**

- > The control opens the **Dyna. Coll. Monitoring (DCM)** window.
- Activate DCM in the desired operating modes, using the toggle switches
- Press **OK**

- > The control activates DCM in the selected operating modes.

The control displays the status of Dynamic Collision Monitoring (DCM) in the **Positions** workspace. When deactivating DCM, the control displays an icon in the information bar.

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.

11.2 Fixture management

11.2.1 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 246
- Integrating an STL file as workpiece blank

  Further information: Programming and Testing User's Manual

Requirements

- Kinematics description

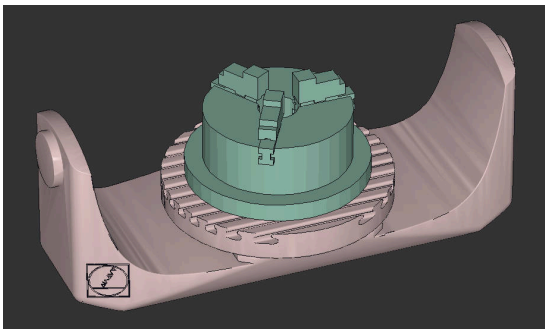
  The machine manufacturer creates the kinematics description
- Insertion point defined

  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 254
- Fixture placement
  - The **Set up fixtures** function in the **Setup** (#140 / #5-03-2) application
    Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256
  - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
  Further information: Programming and Testing User's Manual



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: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356

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: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356

Fixtures from CFG files

CFG files are configuration files. You can integrate the STL and M3D files available in a CFG file. This enables you to map complex setups.

The **Set up fixtures** function can be used to create a CFG file for the fixture, using the measured value.

In CFG files, you can correct the orientation of the fixture files to be in effect on the control. **KinematicsDesign** can be used to create and edit CFG files on the control.

Further information: "Editing CFG files with KinematicsDesign", Page 266

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: Programming and Testing User's Manual

11.2.2 Integrating fixtures into collision monitoring (#140 / #5-03-2)

Application

The **Set up fixtures** function determines the position of a 3D model in the **Simulation** workspace, matching the real fixture in the machine envelope. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- The **Simulation** workspace
  Further information: Programming and Testing User's Manual
- Dynamic Collision Monitoring (DCM)
  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
- Fixture monitoring
  Further information: "Fixture management", Page 253
- Setting up a workpiece with graphical support (#159 / #1-07-1)
  Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398

Requirements

- Collision Monitoring v2 (#140 / #5-03-2) software option
- Workpiece touch probe
- Permitted fixture file matching the real fixture
  Further information: "Options for fixture files", Page 254

Description of function

The **Set up fixtures** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

The **Set up fixtures** function determines the fixture position using various probing processes. First, one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to improve positioning accuracy. After defining the position in one axis direction, the control changes the status of that axis from red to green.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.

Further information: "Error estimate diagram", Page 261

The scope of the **Set up fixtures** function depends on the Adv. Function Set 1 (#8 / #1-01-1) and Adv. Function Set 2 (#9 / #4-01-1) software options as follows:

- Both software options enabled:

  You can tilt before probing, and incline the tool while probing, in order to probe even complex fixtures.
- Only Adv. Function Set 1 (#8 / #1-01-1) is enabled:

  You can tilt before probing. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.

  > If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.
- None of the two software options is enabled:

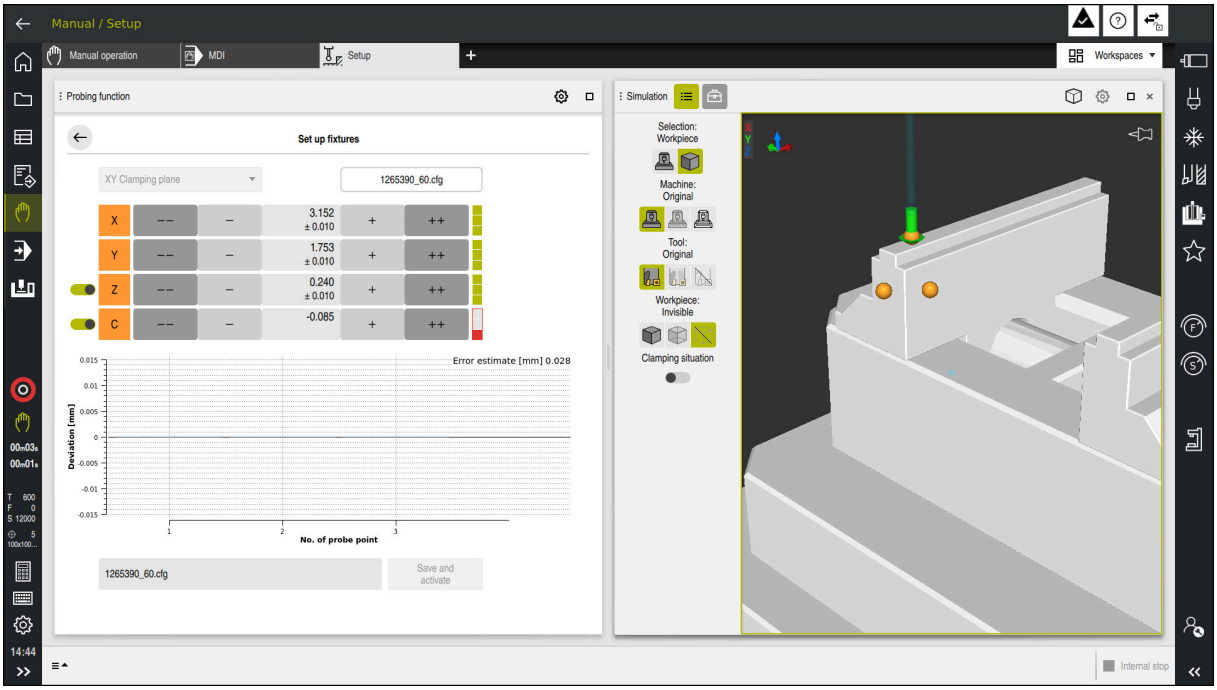
  You cannot tilt before probing. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 235

Further information: Programming and Testing User's Manual

Extension of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphic support for setting up the fixture.



The **Set up fixtures** function with the **Simulation** workspace open

When the **Set up fixtures** function is active, the **Simulation** workspace shows the content below:

- Current position of fixture as viewed by the control
- Probed points on the fixture
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the fixture or the workpiece touch probe is positioned within the fixture, as seen by the control.

    In this case, you can adjust the position of the 3D model in the simulation, if applicable.
  - Red arrow

    Probing in the direction of the arrow is not possible.

    > Probing on edges, corners or heavily curved fixture areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    Probing in the direction of the arrow is possible under certain conditions. Probing is done in a deselected direction or might cause collisions.
  - Green arrow

    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Meaning
XY Clamping plane	This selection menu defines the plane in which the fixture is in contact with the machine. The control offers the following planes: ■ XY clamping plane ■ XZ clamping plane ■ YZ clamping plane **Depending on the selected clamping plane, the control displays the corresponding axis directions. In the **XY Clamping plane**, for example, the control displays the axes **X**, **Y**, **Z** and **C**.**
1265390_60.cfg	Name of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited before saving.
– –	Shifts the position of the virtual fixture by 10 mm, 0.3937 inch, or 10° in the negative axis direction **Fixtures are shifted by mm or inch units in a linear axis and by degrees in a rotary axis.**
–	Shifts the position of the virtual fixture by 1 mm, 0.0394 inch, or 1° in the negative axis direction
-15.982 ± 0.017	■ Enter the position of the virtual fixture directly ■ Value and estimated accuracy after probing
+	Shifts the position of the virtual fixture by 1 mm, 0.0394 inch, or 1° in the positive axis direction
+ +	Shifts the position of the virtual fixture by 10 mm, 0.3937 inch, or 10° in the positive axis direction
	Status of the axis: ■ Dimmed The axis direction is deselected for this set-up process and will not be taken into account. ■ Empty No probing points have been determined yet. ■ Red The control cannot determine the fixture position in this axis direction. ■ Yellow The position of the fixture in this axis direction already contains information. The information is not meaningful yet. ■ Green The control can determine the fixture position in this axis direction.

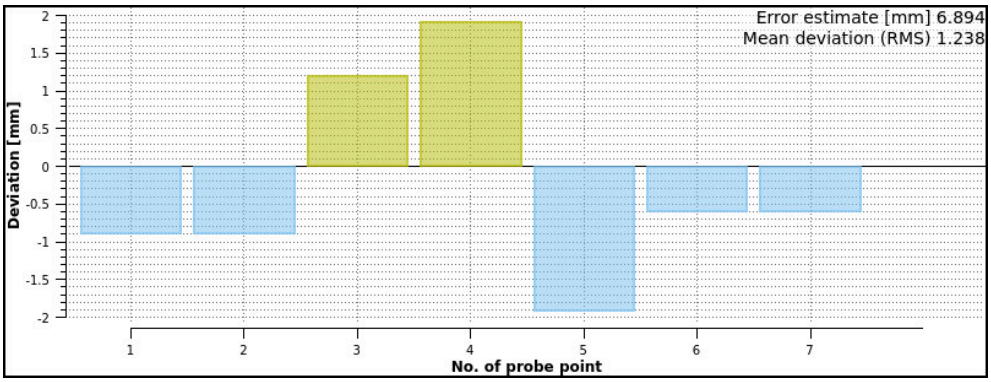
Icon or button	Meaning
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM). When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by **Save and activate** at the end of the measuring process. When creating a new CFG file, enter a different file name next to the button.

When using a datum clamping system and for this reason you do not want to consider one axis direction (such as **Z**) when setting up the fixture, the axis in question can be deselected by a toggle switch. The control will not take deselected axis directions into account in the set-up process and positions the fixture by considering the remaining axis directions only.

Error estimate diagram

Every probing point further restricts the possible positioning of the fixture and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.



Error estimate diagram of the **Set up fixtures** function with transparent columns

The error estimate diagram of the **Set up fixtures** function displays the following information:

- **Error estimate [mm]**

  This value indicates the greatest estimated distance between the 3D model and the fixture after each probing point.
- **Mean deviation (RMS)**

  This value indicates the average of all measured distances between the 3D model and the fixture.
- **Deviation [mm]**

  On this axis, you can see how great the estimated distance between the 3D model and the probing point at the fixture is.
- **No. of probe point**

  This axis shows the number of probing points.
- Columns

  If the status is not yet green for all axes, the control will display transparent columns.

  After each probing point, the control will adjust the 3D model accordingly. This means that the previous values will change, too.

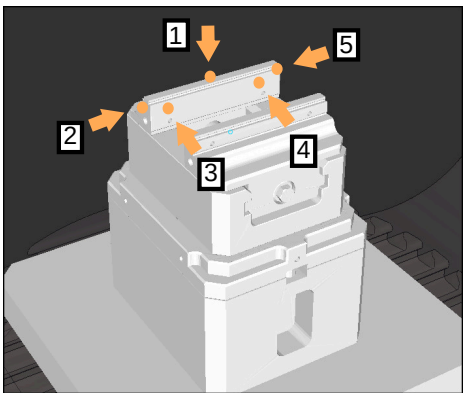
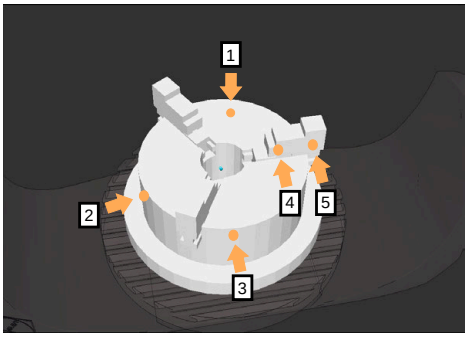
  Once the columns in the error estimate diagram are no longer transparent and the **Error estimate [mm]** value displays the required precision, the set-up process is completed.

The factors below influence the accuracy that can be achieved when measuring fixtures:

- Accuracy of workpiece touch probe
- Repeatability of workpiece touch probe
- Accuracy of 3D model
- Condition of the actual fixture (e.g., existing wear or score marks)

Example of sequence of fixture probing points

Below are some of the probing points that can be set for different fixtures:

Chucking equipment/fixtures	Possible sequence
 Probing points for a vise with a fixed vise jaw	The following probing points can be set when measuring a vise: 1 Touching the fixed vise jaw in **Z-** 2 Touching the fixed vise jaw in **X+** 3 Touching the fixed vise jaw in **Y+** 4 Touching the second value in **Y+** for rotation 5 To improve accuracy, touching the check point in **X-**
 Probing points with a three-jaw chuck	The following probing points can be set when measuring a three-point chuck: 1 Touching the jaw chuck body in **Z-** 2 Touching the jaw chuck body in **X+** 3 Touching the jaw chuck body in **Y+** 4 Touching the jaw in **Y+** for rotation 5 Touching the second value at the jaw in **Y+** for rotation

Measuring the fixed-jaw vise

> The desired 3D model must meet the requirements of the control.
> **Further information:** "Options for fixture files", Page 254

To measure a vise using the **Set up fixtures** function:

- Affix a real vise in the working space

- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the fixed vise jaw at a notable point

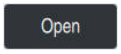
> This step makes the subsequent steps easier.

- Select the **Setup** application

- Select **Set up fixtures**

> The control opens the **Set up fixtures** menu.

- Select a 3D model matching the real vise



- Select **Open**

> The control opens the selected 3D model in the simulation.

- Pre-position the 3D model by using the buttons for the individual axes within the virtual working space

> For pre-positioning the vise, use the workpiece touch probe as a point of reference.
>
> At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using, for example, the table's T-slots produces values close to the position of the real vise.
>
> Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

- Specify the clamping plane (e. g., **XY**)
- Position the workpiece touch probe until a green down arrow appears

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the fixture will actually be touched. Check if the fixture position in the simulation and in the machine match and if touching in the direction of the arrow is possible on the machine.
>
> Do not touch directly near edges, chamfers and roundings.

- Press the **NC Start** key
> The control probes in the direction of the arrow.
> The control displays the status of the **Z** axis in green and shifts the fixture to the touched position. The control marks the touched position by a point in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
> The status of the axes turns green.
- Touch another point in axis direction **Y+** for the basic rotation

> To achieve maximum accuracy when touching the basic rotation, the probing points should be as far apart from one another as possible.

> The control changes the status of the **C** axis to green.
- Touch the check point in axis direction **X-**

> Additional check points at the end of the measuring process improve the matching accuracy and minimize the faults between the 3D model and the real fixture.

Save and activate

- Select **Save and activate**
> The control closes the **Set up fixtures** function, saves a CFG file with the measured values at the path specified above, and integrates the measured fixture into Dynamic Collision Monitoring (DCM).

Notes

NOTICE

Danger of collision!

To probe the clamping situation in the machine exactly, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision.

- Calibrate the workpiece touch probe at regular intervals
- Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the real fixture.
- At the time of set-up, Dynamic Collision Monitoring (DCM) does not know the exact position of the fixture. In this condition, collisions with the fixture, the tool or other non-machine components such as fixing clamps in the work envelope may occur. The non-machine components can be modeled on the control using a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 266
- If you cancel the **Set up fixtures** function, DCM will not monitor the fixture. In this case, any fixtures previously set up are also removed from the scope of monitoring. The control displays a warning.
- Only one fixture can be measured at a time. To monitor several fixtures simultaneously by DCM, the fixtures must be integrated into a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 266
- When measuring a jaw chuck, the coordinates of the axes **Z**, **X** and **Y** are determined just as when measuring a vise. The rotation is determined from one single jaw.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The saved fixture file can be integrated into the NC program with the **FIXTURE SELECT** function. This can be used for simulating and executing the NC program, considering the real setup situation.

  Further information: Programming and Testing User's Manual

11.2.3 Editing CFG files with KinematicsDesign

Application

KinematicsDesign allows editing CFG files in the control. In this process, **KinematicsDesign** displays the fixtures graphically and thus supports troubleshooting and removal of errors.

Related topics

- Combine fixtures into complex clamping arrangements

  Further information: "Combining fixtures in the New Fixture window", Page 271

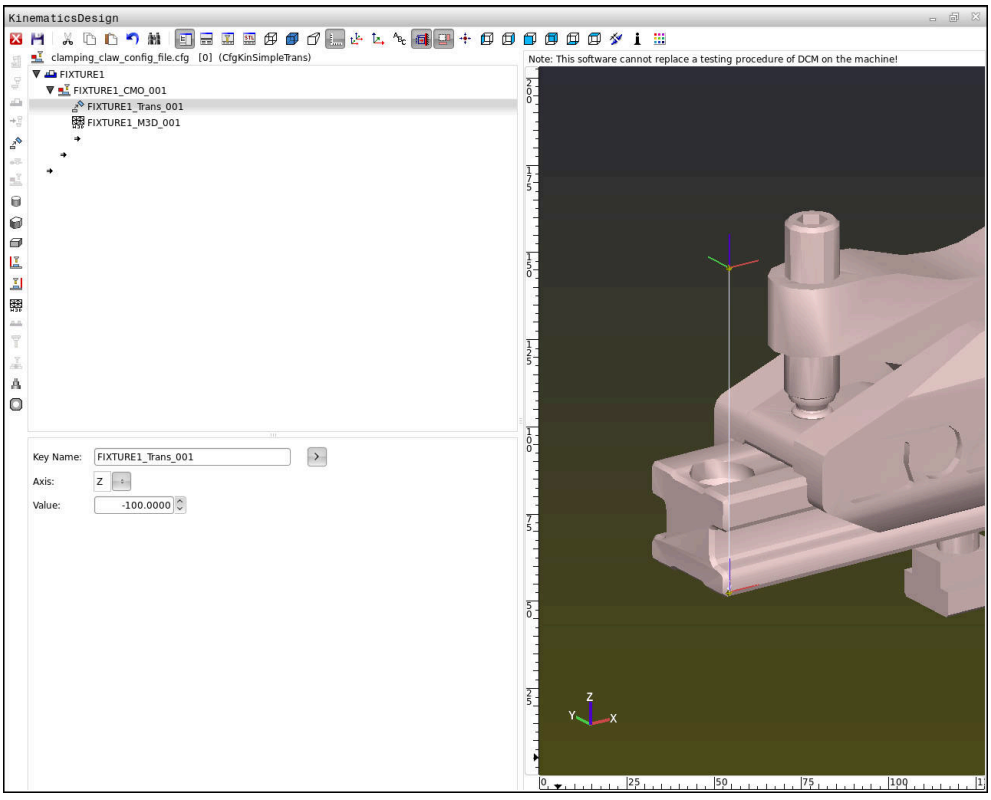
Description of function

When opening a CFG file in the control, the control makes **KinematicsDesign** available as a selection item.

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
  - 3D model (M3D or STL files)
  - Cylinder
  - Prism
  - Cuboid
  - Truncated cone
  - Hole

You can integrate both STL files and M3D files into CFG files more than once.



Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Function	Description
`key:= ""`	Name of the function
`dir:= ""`	Direction of a transformation (e.g., **X**)
`val:= ""`	Value
`name:= ""`	Name displayed if a collision occurs (optional input)
`filename:= ""`	File name
`vertex:= [ ]`	Position of a cube
`edgeLengths:= [ ]`	Dimensions of a cuboid
`bottomCenter:= [ ]`	Center of a cylinder
`radius:=[ ]`	Radius of a cylinder
`height:= [ ]`	Height of a geometric object
`polygonX:= [ ]`	Line of a polygon in X
`polygonY:= [ ]`	Line of a polygon in Y
`origin:= [ ]`	Starting point of a polygon

Each element is assigned its own **key**. A **key** must be unambiguous and unique, meaning that it must not occur more than once in the description of a fixture. Based on the **key**, the elements are referenced to each other.

The following functions are available if you wish to use CFG functions to describe a fixture in the control:

Function	Description
`CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL",name:="")`	Definition of fixture component You can also enter an absolute path for the defined fixture component (e.g., **TNC:\nc_prog\1.STL**)
`CfgKinSimpleTrans(key:="XShiftFixture", dir:=X,val:=0)`	Shift in X axis Inserted transformations, such as a shift or rotation, are effective for all of the elements following in the kinematic chain.
`CfgKinSimpleTrans(key:="CRot0", dir:=C,val:=0)`	Rotation in C axis
`CfgCMO ( key:="fixture", primitives:= ["XShiftFixture","CRot0", "Fixture_body"], active :=TRUE, name :="")`	Describes all of the transformations contained in the fixture. The parameter active **:= TRUE** activates collision monitoring for the fixture. The **CfgCMO** contains collision objects and transformations. The fixture is combined based on the arrangement of the different transformations. Here, the transformation **XShiftFixture** shifts the center of rotation of the transformation **CRot0**.

Function	Description
`CfgKinFixModel(key:="Fix_Model", kinObjects:=["fixture"])`	Fixture designation **CfgKinFixModel** contains one or more **CfgCMO** elements.

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

The following geometric objects are available:

Function	Description
`CfgCMOCuboid ( key:="FIXTURE_Cub", vertex:= [ 0, 0, 0 ], edgeLengths:= [0, 0, 0], name:="" )`	Definition of a cuboid
`CfgCMOCylinder ( key:="FIXTURE_Cyl", dir:=Z, bottomCenter:= [0, 0, 0], radius:=0, height:=0, name:="")`	Definition of a cylinder
`CfgCMOPrism ( key:="FIXTURE_Pris_002", height:=0, polygonX:=[], polygonY:=[], name:="", origin:= [ 0, 0, 0 ] )`	Definition of a prism A prism is described by entering the height and several polygonal lines.

Creating a fixture entry with a collision object

The content below describes the procedure with **KinematicsDesign** opened.

To create a fixture entry with a collision object:

- ▶ Select **Insert chucking equipment**
- > **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a **keyname** for the fixture (e.g., **clamping jaw**)
- ▶ Confirm your input
- > **KinematicsDesign** loads the input.

- ▶ Move cursor down one level

- ▶ Select **Insert collision object**
- ▶ Confirm your input
- > **KinematicsDesign** creates a new collision object.

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:

- Create a fixture entry with a collision object

- Select the cursor key beneath the collision object

- Select the desired geometric shape (e.g., a cuboid)
- Define the position of the cuboid (e.g., **X = 0**, **Y = 0**, **Z = 0**)
- Define the dimensions of the cuboid (e.g., **X = 100**, **Y = 100**, **Z = 100**)
- Confirm your input
- > The control displays the defined cuboid in the graphic.

Integrating 3D models

The integrated 3D models must meet the requirements of the control.

To integrate a 3D model as a fixture:

- Create a fixture entry with a collision object

- Select the cursor key beneath the collision object

- Select **Insert 3D model**
- > The control opens the **Open File** window.
- Select the desired STL or M3D file
- Press **OK**
- > The control integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

- Define the fixture

- Select the cursor key beneath the element to be positioned

- Select **Insert transformation**
- Enter a **key name** for the transformation (e.g., **Z shift**)
- Select the **axis** for the transformation (e.g., **Z**)
- Select the **value** for the transformation (e.g., **100**)
- Confirm your input
- > **KinematicsDesign** inserts the transformation.
- > **KinematicsDesign** depicts the transformation in the graphic.

Notes

- If one of the transformations contains the **?** character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

  Further information: "Combining fixtures in the New Fixture window", Page 271
- As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
`CfgCMOMesh3D (key:="Fixture_body", filename:="vise_47155.STL", name:="")`	Body of the vise
`CfgCMOMesh3D (key:="vise_jaw_1", filename:="vise_jaw_47155.STL", name:="")`	First jaw of the vise
`CfgCMOMesh3D (key:="vice_jaw_2", filename:="vise_jaw_47155.STL", name:="")`	Second jaw of the vise

Definition of jaw opening width

In this example, the opening width of the vise is defined using two mutually dependent transformations.

Code	Explanation
`CfgKinSimpleTrans (key:="TRANS_opening_width", dir:=Y, val:=-60)`	Jaw opening width of the vise in Y direction: 60 mm
`CfgKinSimpleTrans (key:="TRANS_opening_width_2", dir:=Y, val:=30)`	Position of the first jaw of the vise in Y direction: 30 mm

Positioning of the fixture within the working space

The defined fixture components are positioned using various transformations.

Code	Explanation

`CfgKinSimpleTrans (key:="TRANS_X", dir:=X, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Y", dir:=Y, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z", dir:=Z, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z_vise_jaw", dir:=Z, val:=60)`
`CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180)`
`CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)`
	Positioning of the fixture components
	In this example, a rotation by 180° is inserted for rotating the defined jaw of the vise. This is necessary because the same initial model is used for both jaws of the vise.
	The rotation inserted applies to all subsequent components in the transformation chain.

Description of the fixture

You need to combine all objects and transformations in the CFG file in order to ensure that the fixture is correctly depicted in the simulation.

Code	Explanation

`CfgCMO (key:="FIXTURE", primitives:= [`
`"TRANS_X",`
`"TRANS_Y",`
`"TRANS_Z",`
`"TRANS_SPC",`
`"TRANS_SPB",`
`"TRANS_SPA",`
`"Fixture_body",`
`"TRANS_Z_vise_jaw",`
`"TRANS_opening_width_2",`
`"vicse_jaw_1",`
`"TRANS_opening_width",`
`"TRANS_C_180",`
`"vise_jaw_2" ], active:=TRUE, name:="")`
	Combining the transformations and objects contained in the fixture

Fixture designation

You need to assign a designation to the combined fixture.

Code	Explanation
`CfgKinFixModel (key:="FIXTURE1",` ` kinObjects:=["FIXTURE"])`	Designation of the combined fixture

11.2.4 Combining fixtures in the New Fixture window

Application

The **New Fixture** window allows combining several fixtures and saving them as a new fixture. This enables realizing and monitoring complex clamping situations.

Related topics

- Fundamentals of fixtures
  Further information: "Fundamentals", Page 253
- Integrating fixtures into the NC program
  Further information: Programming and Testing User's Manual
- Set up fixtures (#140 / #5-03-2)
  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256

Requirement

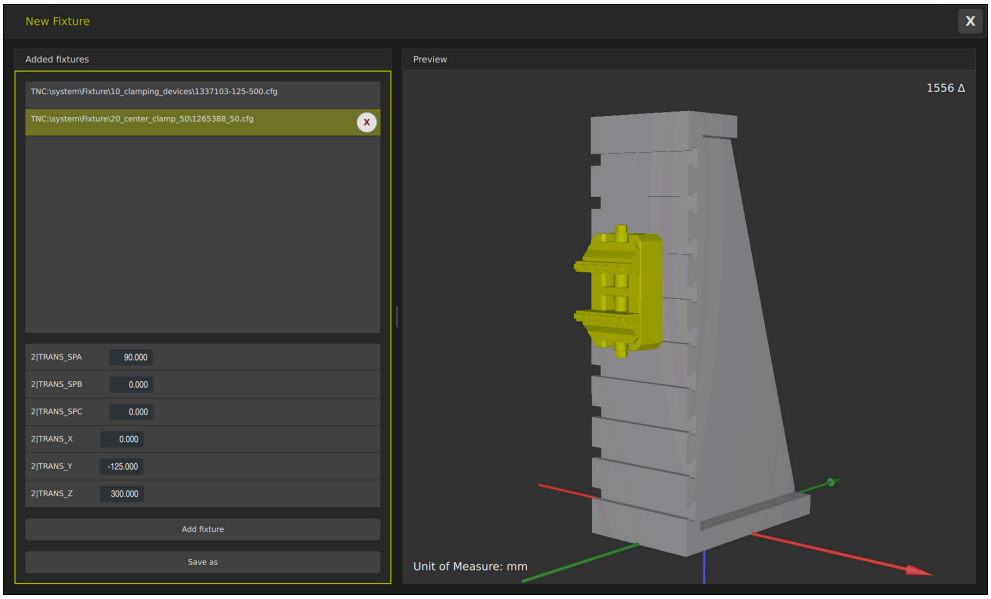
- Fixtures of suitable format:
  - STL file
    - 20,000 triangles maximum
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

Description of function

To navigate to this function:

Tools ▶ **Combine fixtures**

The control also makes this function available as a selection option for opening CFG files.



Combined fixture with variable transformations

The **Add fixture** button selects all required fixtures one by one.

If one of the transformations contains the **?** character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

The control displays a preview of the combined fixture and the total number of all triangles.

The **Save as** button saves the combined fixture as a CFG file.

Notes

- For optimum performance, HEIDENHAIN recommends that combined fixtures contain no more than 20,000 triangles.
- If the position or the size of a fixture must be adapted, use **KinematicsDesign**.
  Further information: "Editing CFG files with KinematicsDesign", Page 266

12

Control Functions

12.1 Adaptive Feed Control (AFC) (#45 / #2-31-1)

12.1.1 Fundamentals

Application

Adaptive Feed Control (AFC) saves time when processing NC programs and reduces wear on the machine. The control regulates the contouring feed rate during program run depending on the spindle power. In addition, the control responds to overloading of the spindle.

Related topics

- Tables related to AFC
  Further information: "Tables for AFC (#45 / #2-31-1)", Page 512

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: "Basic AFC settings in AFC.tab", Page 512
- Define settings for AFC for each tool in the tool management
  Further information: "Tool table tool.t", Page 457
- Define AFC in the NC program
  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 279
- 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 280
- Prior to automatic control, determine the reference spindle power with a teach-in cut
  Further information: "AFC teach-in cut", Page 282

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

Further information: "The Positions workspace", Page 119

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

Further information: "The AFC tab (#45 / #2-31-1)", Page 129

Benefits of AFC

Adaptive feed control (AFC) has the following advantages:

- Optimization of machining time

  By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring

  If the spindle power exceeds the taught-in or specified maximum value, the control reduces the feed until the reference spindle power is reached. If the minimum feed rate is exceeded, the control executes a shutdown response. AFC can also use the spindle power to monitor the tool for wear and breakage without changing the feed rate.

  Further information: "Monitoring tool wear and tool load", Page 284
- Protection of the machine's mechanical elements

  Timely feed rate reduction and shutdown reactions help to avoid machine overload.

Tables related to AFC

The control offers the following tables in conjunction with AFC:

- **AFC.tab**

  In the **AFC.tab** table, you define the feed-rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

  Further information: "Basic AFC settings in AFC.tab", Page 512
- ***.H.AFC.DEP**

  With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

  Further information: "AFC.DEP settings file for teach-in cuts", Page 515
- ***.H.AFC2.DEP**

  During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you are perfoming the teach-in cut.

  In control mode, the control updates the data in this table and performs evaluations.

  Further information: "Log file AFC2.DEP", Page 516

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of the active NC program.

Further information: "Editing the tables for AFC", Page 517

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as Adaptive Feed Control (AFC) is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g., due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This behavior applies regardless of how the function is deactivated. This feed acceleration may result in damage to the tool and/or the workpiece!

- If the feed rate is about to fall below the **FMIN** value, stop the machining operation (instead of deactivating the AFC function)
- Define the overload response for cases in which the feed rate falls below the **FMIN** value

- If Adaptive Feed Control is active in **Control** mode, the control executes a shutdown response independent of the programmed overload response.
  - If, with the reference spindle load, the value falls below the minimum feed factor

    The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.

    Further information: "Basic AFC settings in AFC.tab", Page 512
  - 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: "Monitoring tool wear and tool load", Page 284
- 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: Programming and Testing User's Manual

12.1.2 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: "Basic AFC settings in AFC.tab", Page 512
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 457
- **AFC** toggle switch active

  Further information: "The AFC toggle switch in the Program Run operating mode", Page 280

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 279

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: "The Positions workspace", Page 119

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.

12.1.3 AFC teach-in cut

Fundamentals

Application

With the teach-in cut, the control determines the reference power of the spindle for the machining step. Based on the reference power, the control adjusts the feed rate in control mode.

If you have already determined the reference power for a machining operation, you can specify the value for the machining operation. For this, the control provides the **AFC-LOAD** column in the tool management and the **LOAD** syntax element in the **FUNCTION AFC CUT BEGIN** function. In this case, the control no longer performs a teach-in cut, but uses the specified value immediately for control.

Related topics

- Enter the known reference power in the **AFC-LOAD** column in the tool management
  Further information: "Tool table tool.t", Page 457
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function
  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 279

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Control settings defined in the **AFC.tab** table
  Further information: "Basic AFC settings in AFC.tab", Page 512
- Desired control setting defined for all tools
  Further information: "Tool table tool.t", Page 457
- Desired NC program selected in the **Program Run** operating mode
- **AFC** toggle switch active
  Further information: "The AFC toggle switch in the Program Run operating mode", Page 280

Description of function

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**.

Further information: "AFC.DEP settings file for teach-in cuts", Page 515

When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

When the control has determined the control reference power, it ends the teach-in cut and switches to control mode.

Notes

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.
- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.
- If the determined reference load is greater than 2%, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.
- In **FUNCTION MODE TURN** machining mode, the minimum reference load is 5%. Even if the control determines lower values, it will still use this minimum reference load. Thus, the overload limits (indicated as percentage values) are based on a minimum reference load of at least 5%.

The AFC settings button

Application

The **AFC settings** button in the **Program Run** operating mode allows terminating a teach-in cut or opening the tables for AFC.

Related topics

- Fundamentals for the teach-in cut
  Further information: "Fundamentals", Page 282
- Tables for AFC
  Further information: "Tables for AFC (#45 / #2-31-1)", Page 512

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

This button offers the following select options:

Button	Meaning
AFC.TAB	Editing the factory default settings When selecting this button, the control will open the **AFC.TAB** table in the **Tables** operating mode. **Further information:** "Basic AFC settings in AFC.tab", Page 512
AFC.DEP	Editing the settings file for teach-in cuts When selecting this button, the control will open the **AFC.DEP** table for the current NC program in the **Tables** operating mode. **Further information:** "AFC.DEP settings file for teach-in cuts", Page 515
AFC2.DEP	Editing the log file for evaluation When selecting this button, the control will open the **AFC2.DEP** table for the current NC program in the **Tables** operating mode. **Further information:** "Log file AFC2.DEP", Page 516
Stop Teach	Terminating a teach-in cut ■ The control terminates the teach-in cut and changes to control mode **Further information:** "AFC teach-in cut", Page 282 ■ In the **AFC.DEP** table, the control changes the status of the **ST** column from teaching-in (**L**) to controlling (**C**). **Further information:** "AFC.DEP settings file for teach-in cuts", Page 515 ■ In the **Positions** workspace, the control changes the icon for the teaching-in cut into the control mode icon. **Further information:** "The Positions workspace", Page 119

> In a milling operation, you do not have to run the entire machining step in teaching-in mode. If the cutting conditions do not change significantly, you can switch to control mode immediately.

12.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear or breakage. To do this, use columns **AFC-OVLD1** or **AFC-OVLD2** in the tool management.

The control offers tool wear and tool load monitoring even in turning mode (#50 / #4-03-1).

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management
  Further information: "Tool table tool.t", Page 457

Description of function

If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active.

Further information: "Basic AFC settings in AFC.tab", Page 512

Tool wear and tool breakage cannot be monitored at the same time. If the **AFC_OVLD2** column contains a value, the control will ignore the **AFC_OVLD1** column.

Tool wear monitoring

Activate cut-related tool wear monitoring by entering a value not equal to 0 in the **AFC-OVLD1** column in the tool table.

The overload response depends on the **AFC.TAB** column **OVLD**.

In conjunction with cut-related tool wear monitoring, the control only evaluates the options **M**, **E**, and **L** in the **OVLD** column. The following responses are possible:

- Pop-up window
- Lock current tool
- Insert replacement tool

Tool load monitoring

Activate cut-related tool load monitoring (tool breakage control) by entering a value not equal to 0 in the **AFC-OVLD2** column in the tool table.

As overload response, the control always executes a machining stop and locks the momentary tool.

In turning mode, the control can check for tool wear and tool breakage.

Tool breakage leads to a sudden load decrease. If you want the control to monitor the load decrease, too, enter the value 1 in the SENS column.

Further information: "Basic AFC settings in AFC.tab", Page 512

Example

The entries in columns **AFC-OVLD1** and **AFC-OVLD2** are added to the feedback-control reference power **AFC-LOAD**.

Further information: "AFC teach-in cut", Page 282

Example input for tool wear and tool load monitoring:

Column	Input
AFC-LOAD	30%
AFC-OVLD1	5%
AFC-OVLD2	10%

In this example, the control adds the 5% and 10% to the 30% in each case.

As soon as a value is defined in column **AFC-OVLD1**, the tool will monitor tool wear. When the control used in the example reaches a spindle power of 35% in total, it executes the defined reaction.

12.2 Active Chatter Control (ACC) (#145 / #2-30-1)

Application

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates.

Related topics

- **ACC** column in the tool table
  Further information: "Tool table tool.t", Page 457

Requirements

- Software option Active Chatter Contr. (#145 / #2-30-1)
- Control adapted by the machine manufacturer
- **ACC** column in the tool management defined with **Y**
- Number of tool cutting edges defined in the **CUT** column

Description of function

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25%. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

ACC is activated and deactivated using the **ACC** toggle switch in the **Program Run** operating mode or the **MDI** application.

Further information: "The Program Run operating mode", Page 412

Further information: "The MDI Application ", Page 369

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

Further information: "The Positions workspace", Page 119

Notes

- ACC reduces or prevents vibrations in the range of 20 Hz to 150 Hz. If ACC does not appear to have an effect, the vibrations may be outside of this range.
- With the software option Machine Vibr. Contr. (#146 / #2-24-1), you can additionally improve the result.

12.3 Global Program Settings GPS (#44 / #1-06-1)

12.3.1 Fundamentals

Application

The Global Program Settings (GPS) allow you to define selected transformations and settings without changing the NC program. All of the settings apply globally and are superimposed on the relevant active NC program.

Related topics

- Coordinate transformations in the NC program
  Further information: Programming and Testing User's Manual
  Further information: User's Manual for Machining Cycles
- The **GPS** tab in the **Status** workspace
  Further information: "The GPS tab (#44 / #1-06-1)", Page 133
- Reference systems of the control
  Further information: "Reference systems", Page 214

Requirement

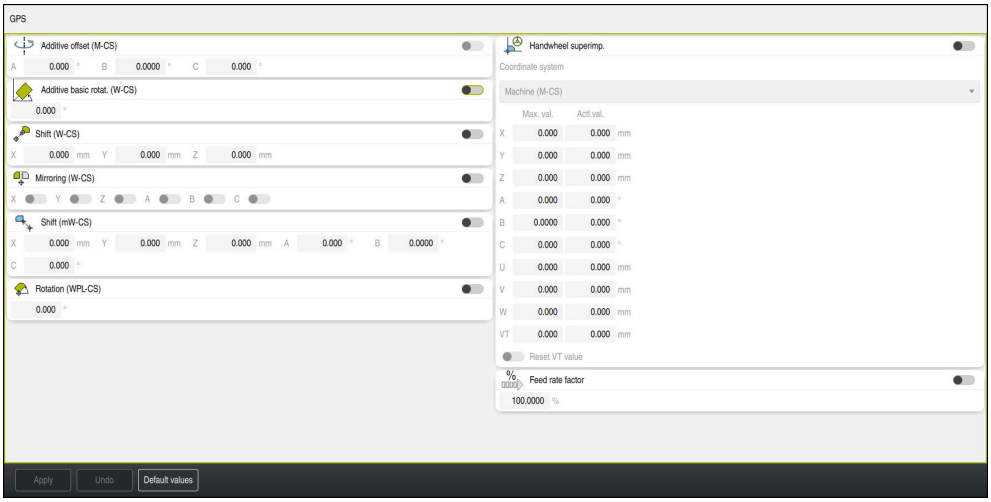
- Software option Global PGM Settings (#44 / #1-06-1)

Description of function

The values of the Global Program Settings are defined and activated in the **GPS** workspace.

The **GPS** workspace is available in the **Program Run** operating mode and in the **MDI** application of the **Manual** operating mode.

The transformations of the **GPS** workspace are effective in all operating modes and are persistent across reboots of the control.



The **GPS** workspace with active functions

The functions of GPS are activated using toggle switches.

The control marks the sequence in which the transformations are effective with green digits.

The control shows the active GPS settings on the **GPS** tab of the **Status** workspace.

Further information: "The GPS tab (#44 / #1-06-1)", Page 133

Before executing an NC program with active GPS in the **Program Run** operating mode, you must confirm use of the GPS functions in a pop-up window.

Buttons

The control provides the following buttons in the **GPS** workspace:

Button	Description
Apply	Save changes in the **GPS** workspace
Undo	Reset unsaved changes in the **GPS** workspace
Default values	Set the **Feed rate factor** function to 100%, reset all other functions to zero

Overview of Global Program Settings (GPS)

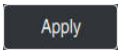
The Global Program Settings (GPS) include the following functions:

Function	Description
Additive offset (M-CS)	Shift of the zero position of an axis in the machine coordinate system **M-CS** **Further information:** "The Additive offset (M-CS) function", Page 290
Additive basic rotat. (W-CS)	Additional rotation based on basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. **Further information:** "The Additive basic rotat. (W-CS) function", Page 292
Shift (W-CS)	Shift of workpiece preset in a single axis in the workpiece coordinate system **W-CS** **Further information:** "The Shift (W-CS) function", Page 292
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system **W-CS** **Further information:** "The Mirroring (W-CS) function", Page 293
Shift (mW-CS)	Additional shift of a workpiece datum already shifted in the modified workpiece coordinate system **(mW-CS)**. **Further information:** "The Shift (mW-CS) function", Page 294
Rotation (WPL-CS)	Rotation around the active tool axis in the working plane coordinate system **WPL-CS** **Further information:** "The Rotation (WPL-CS) function", Page 296
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel **Further information:** "The Handwheel superimp. function", Page 296
Feed rate factor	Manipulation of the active feed rate **Further information:** "The Feed rate factor function ", Page 299

Defining and activating Global Program Settings (GPS)

To define and activate the Global Program Settings (GPS):

- Select an operating mode (e.g., **Program Run**)
- Open the **GPS** workspace
- Activate the toggle switch of the desired function (e.g., **Additive offset (M-CS)**)
- > The control activates the selected function.
- Enter a value in the desired field (e.g., **A=10.0°**)



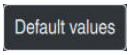
- Press **Apply**
- > The control accepts the entered values.

> If you select an NC program for program run, you must confirm the Global Program Settings (GPS).

Resetting Global Program Settings (GPS)

To reset the Global Program Settings (GPS):

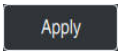
- Select an operating mode (e.g., **Program Run**)
- Open the **GPS** workspace



- Select **Default values**

> Provided that you have not selected the **Apply** button, you can restore the values with the **Undo** function.

- > The control sets the values of all Global Program Settings (GPS) to zero except for the feed factor.
- > The control sets the feed factor to 100%.



- Press **Apply**
- > The control saves the values that have been reset.

Notes

- The control dims any axes that are not active on your machine.
- Value inputs are defined in the selected unit of measurement for the position display (mm or inch units). These values include offset values and values of **Handwheel superimp.**. Angles are always entered in degrees.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The use of touch-probe functions deactivates the Global Program Settings (GPS) (#44 / #1-06-1) temporarily.
- The optional machine parameter **CfgGlobalSettings** (no. 128700) can be used to define which GPS functions are available on the control. The machine manufacturer enables this parameter.

12.3.2 The Additive offset (M-CS) function

Application

With the **Additive offset (M-CS)** function, you can shift the zero position of a machine axis in the machine coordinate system **M-CS**. You can use this function, for example, on large machines, to correct an axis when using axis angles.

Related topics

- Machine coordinate system **M-CS**
  Further information: "Machine coordinate system M-CS", Page 216
- Difference between basic rotation and offset
  Further information: "Basic transformation and offset", Page 507

Description of function

The control adds the value to the active axis-specific offset from the preset table.

Further information: "Preset table *.pr", Page 503

If you activate a value in the **Additive offset (M-CS)** function, the zero position of the affected axis changes in the position display of the **Positions** workspace. The control assumes a different zero position of the axes.

Further information: "The Positions workspace", Page 119

Application example

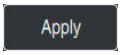
The travel range of a machine with AC fork head is increased using the **Additive offset (M-CS)** function. An eccentric tool chuck is used and the zero position of the C axis is shifted by 180°.

Initial situation:

- Machine kinematics with AC fork head
- Use of an eccentric tool chuck
  The tool is clamped in an eccentric tool chuck outside the center of rotation of the C axis.
- The machine parameter **presetToAlignAxis** (no. 300203) for the C axis is set to **FALSE**

To increase the traversing distance:

- ▶ Open the **GPS** workspace
- ▶ Activate the **Additive offset (M-CS)** toggle switch
- ▶ Enter **C 180°**



- ▶ Press **Apply**
- ▶ Program a positioning movement with **L C+0** in the desired NC program
- ▶ Select an NC program
- > The control considers the 180° rotation for all C axis positioning movements as well as the changed tool position.
- > The position of the C axis does not affect the position of the workpiece preset.

Notes

- After having activated an additive offset, reset the workpiece preset.
- The machine manufacturer uses the optional machine parameter **preset-ToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offsets in the following NC functions:
  - **FUNCTION PARAXCOMP**
  - **POLARKIN** (#8 / #1-01-1)
  - **FUNCTION TCPM** or **M128** (#9 / #4-01-1)
  - **FACING HEAD POS** (#50 / #4-03-1)
    Further information: Programming and Testing User's Manual

12.3.3 The Additive basic rotat. (W-CS) function

Application

The **Additive basic rotat. (W-CS)** function enables, for example, a better use of the workspace. For example, you can rotate an NC program by 90° so that the X and Y directions are inverted during execution.

Description of function

The **Additive basic rotat. (W-CS)** function takes effect in addition to the basic rotation or 3D basic rotation from the preset table. The values of the preset table do not change in this respect.

Further information: "Preset table *.pr", Page 503

The **Additive basic rotat. (W-CS)** function has no effect on the position display.

Application example

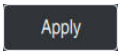
You rotate the CAM output of an NC program by 90° and compensate for the rotation using the **Additive basic rotat. (W-CS)** function.

Initial situation:

- Available CAM output for gantry-type milling machine with a large range of traverse of the Y axis
- The available machining center has the necessary traversing range only in the X axis
- The workpiece blank is clamped with a 90° rotation (long side along the X axis)
- The NC program must be rotated by 90° (algebraic sign depends on the preset position)

To rotate the CAM output:

- ▶ Open the **GPS** workspace
- ▶ Activate the **Additive basic rotat. (W-CS)** toggle switch
- ▶ Enter **90°**



- ▶ Press **Apply**
- ▶ Select NC program
- > The control considers the 90° rotation for all axis positioning movements.

12.3.4 The Shift (W-CS) function

Application

You may use the **Shift (W-CS)** function to, for example, rework in order to compensate for the relative offset of a position that is difficult to probe and the workpiece datum.

Description of function

The **Shift (W-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 220

The **Shift (W-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 149

Application example

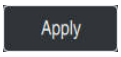
The surface of a workpiece to be reworked is determined using the handwheel and the offset is compensated for using the **Shift (W-CS)** function.

Initial situation:

- Reworking of a free-form surface is required
- Workpiece clamped
- Basic rotation and workpiece preset measured in the working plane
- Z coordinate must be defined with the handwheel due to the presence of a free-form surface

To shift the workpiece surface of a workpiece to be reworked:

- Open the **GPS** workspace
- Activate the **Handwheel superimp.** switch
- Determine the workpiece surface by scratching, using the handwheel
- Activate the **Shift (W-CS)** toggle switch
- Transfer the determined value to the corresponding axis of the **Shift (W-CS)** function (e.g., **Z**)



- Press **Apply**
- Start an NC program
- Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- Determine the workpiece surface by scratching, using the handwheel for fine adjustment
- Select NC program

> The control takes the **Shift (W-CS)** into account.

> The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

12.3.5 The Mirroring (W-CS) function

Application

You can use the **Mirroring (W-CS)** function to execute mirror-inverted execution of an NC program without having to modify the NC program.

Description of function

The **Mirroring (W-CS)** function acts on an axis-by-axis basis. The value is additive to mirroring defined in the NC program before tilting the working plane with Cycle **8 MIRRORING** or the **TRANS MIRROR** function.

Further information: User's Manual for Machining Cycles

Further information: Programming and Testing User's Manual

The **Mirroring (W-CS)** function has no effect on the position display in the **Positions** workspace.

Further information: "Position displays", Page 149

Application example

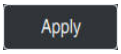
The **Mirroring (W-CS)** function makes the control carry out machining in a mirror-inverted way.

Initial situation:

- A CAM output exists for the non-mirrored workpiece (e.g., for a right-side mirror cap)
- CAM output with the following properties:
  - Output to the tool center point of the ball-nose cutter.
  - **FUNCTION TCPM** defined with **AXIS SPAT** selected
- Workpiece datum positioned at the workpiece blank center

For mirror-inverted machining:

- ▶ Open the **GPS** workspace
- ▶ Activate the **Mirroring (W-CS)** switch
- ▶ Activate the **X** switch



- ▶ Press **Apply**
- ▶ Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

Notes

- If you use **PLANE** functions or the **FUNCTION TCPM** function with spatial angles, the rotary axes are mirrored accordingly along with the mirrored main axes. This always creates the same constellation, regardless of whether the rotary axes were marked in the **GPS** workspace.
- With **PLANE AXIAL**, the mirroring of rotary axes is irrelevant.
- With the **FUNCTION TCPM** function with axis angles, you must activate all axes to be mirrored individually in the **GPS** workspace.

12.3.6 The Shift (mW-CS) function

Application

You can use the **Shift (mW-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult in the modified workpiece coordinate system **mW-CS** , for example.

Description of function

The **Shift (mW-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 220

The **Shift (mW-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 149

A modified workpiece coordinate system **mW-CS** is present with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without these preceding coordinate transformations, the **Shift (mW-CS)** option would be effective directly in the workpiece coordinate system (**W-CS**) and would thus be identical to **Shift (W-CS)**.

Application example

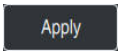
You mirror the CAM output of an NC program. After mirroring, you shift the workpiece datum in the mirrored coordinate system to produce the counterpart to a mirror cap.

Initial situation:

- Available CAM output for a right-side mirror cap
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **Function TCPM** function with spatial angles output
- The left-side mirror cap is to be machined

To shift the datum in the mirrored coordinate system:

- Open the **GPS** workspace
- Activate the **Mirroring (W-CS)** toggle switch
- Activate the **X** toggle switch
- Activate the **Shift (mW-CS)** toggle switch
- Enter the value for shifting the workpiece datum in the mirrored coordinate system



- Press **Apply**
- Run the NC program

> The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

> The control takes the modified position of the workpiece datum into account.

12.3.7 The Rotation (WPL-CS) function

Application

With the **Rotation (WPL-CS)** function, you can, for example, compensate for the misalignment of a workpiece in the already swiveled working plane coordinate system **WPL-CS** without modifying the NC program.

Description of function

The **Rotation (WPL-CS)** function is effective in the tilted working plane coordinate system **WPL-CS**. The value is added to a rotation in the NC program with Cycle **10 ROTATION** or the **TRANS ROTATION** function.

Further information: User's Manual for Machining Cycles

Further information: Programming and Testing User's Manual

The **Rotation (WPL-CS)** function has no effect on the position display.

12.3.8 The Handwheel superimp. function

Application

With the **Handwheel superimp.** function, you can traverse the axes with the superimposed handwheel during program run. You select the coordinate system in which the **Handwheel superimp.** function is effective.

Related topics

- Handwheel superimpositioning with **M118**
  Further information: Programming and Testing User's Manual

Description of function

In the **Max. val.** column, you define the maximum traversing distance for the respective axis. The traverse can be either in the positive or in the negative direction. The maximum path is therefore twice as large as the input value.

In the **Actl.val.** column, the control displays the path traversed using the handwheel for each axis.

The **Actl.val.** column can also be edited manually. If you enter a value greater than the **Max. val.**, you cannot activate the value. The control marks an incorrect value in red. The control displays a warning message and prevents the form from being closed.

If the **Actl.val.** column contains a value when you activate the function, the control will use the menu for returning to move to the new position.

Further information: "Returning to the contour", Page 432

The **Handwheel superimp.** function affects the position display in the **Positions** workspace. The control shows the values offset by the handwheel in the position display.

Further information: "The Positions workspace", Page 119

The control displays the values of the two methods for **Handwheel superimp.** on the **POS HR** tab of the additional status display.

On the **POS HR** tab of the **Status** workspace, the control shows whether the **Max. val.** is defined using the **M118** function or the Global Program Settings (GPS).

Further information: "The POS HR tab", Page 139

Virtual tool axis VT

The virtual tool axis **VT** is needed for machining operations with inclined tools (e.g., for manufacturing oblique holes without using a tilted working plane).

Handwheel superimp. can also be executed in the active tool axis direction. The **VT** always corresponds to the direction of the active tool axis. On machines with head rotation axes, this direction may not correspond to the basic coordinate system **B-CS**. You activate the function with the **VT** line.

Further information: "Notes concerning different machine kinematics", Page 236

By default, values traversed with the handwheel in the **VT** remain active even after a tool change. If you activate the **Reset VT value** toggle switch, the control resets the actual value of the **VT** when a tool is changed.

The control displays the values of the virtual tool axis **VT** on the **POS HR** tab of the **Status** workspace.

Further information: "The POS HR tab", Page 139

For the control to display values, you must define a value greater than 0 in the **VT** function for **Handwheel superimp.**

Notes

NOTICE

Danger of collision!

The coordinate system chosen in the selection menu also takes effect on **Handwheel superimp.** with **M118**, even if the Global Program Settings function (GPS) is not active. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Before exiting the form, always make sure to select the **Machine (M-CS)** coordinate system
- Test the behavior at the machine

NOTICE

Danger of collision!

When both methods for **Handwheel superimp.** with **M118** and with the Global Program Settings GPS are active at the same time, the definitions influence each other, depending on their sequence of activation. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Use only one method for **Handwheel superimp.**
- Preferably use the **Handwheel superimp.** option of the **Global Program Settings** function
- Test the behavior at the machine

HEIDENHAIN does not recommend using both methods for **Handwheel superimp.** at the same time. If **M118** cannot be removed from the NC program, you should at least activate **Handwheel superimp.** from GPS prior to selecting the program. This ensures that the control uses the GPS function rather than **M118**.

- If neither the NC program nor the Global Program Settings were used to activate coordinate system transformations, **Handwheel superimp.** is effective in the same manner in all coordinate systems.
- If, while machining with active Dynamic Collision Monitoring DCM (#40 / #5-03-1), you want to use **Handwheel superimp.**, then the control must be in a stopped or interrupted state. Alternatively, you can also deactivate DCM.

  Further information: "Dynamic Collision Monitoring (DCM) (#40 / #5-03-1)", Page 246
- **Handwheel superimp.** in virtual axis direction **VT** requires neither a **PLANE** function nor the **FUNCTION TCPM** function.
- Use the machine parameter **axisDisplay** (no. 100810) to define whether the control also shows the virtual axis **VT** in the position display of the **Positions** workspace.

  Further information: "The Positions workspace", Page 119

12.3.9 The Feed rate factor function

Application

You can use the **Feed rate factor** function to influence the effective feed rates on the machine (e.g., to adjust the feed rates of a CAM program). This will prevent the CAM program from being re-output using the postprocessor. When doing so, you change all feed rates as a percentage without making any changes in the NC program.

Related topics

- Feed rate limit **F MAX**

  The **Feed rate factor** function has no influence on the feed rate limit with **F MAX**.

  Further information: "Feed rate limit F LIMIT", Page 417

Description of function

All feed rates are changed as a percentage. You define a percentage value from 1% to 1000%.

The **Feed rate factor** function acts on the programmed feed rate and the feed rate potentiometer, but not on rapid traverse **FMAX**.

The control shows the current feed rate in field **F** of the **Positions** workspace. If the **Feed rate factor** function is active, the feed rate is shown with the defined values taken into account.

Further information: "Presets and technology values", Page 121

13

Monitoring

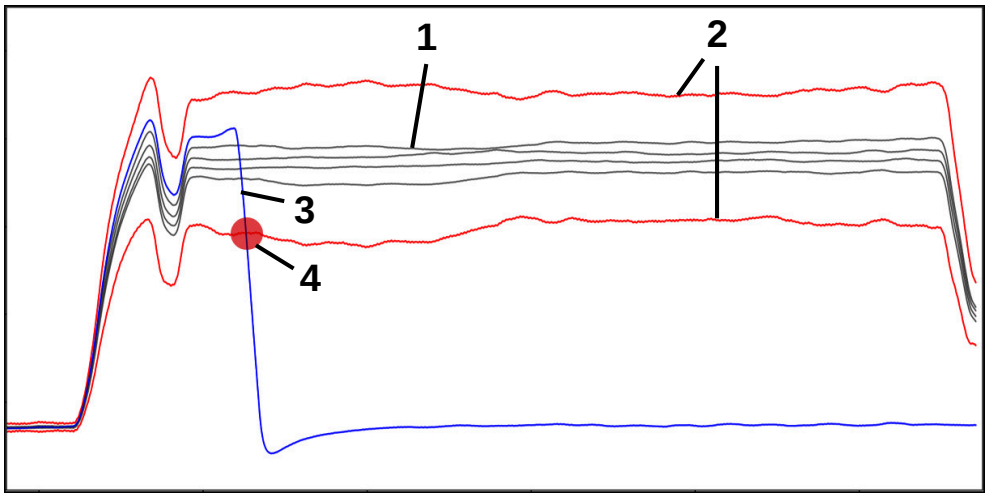
13.1 Process monitoring (#168 / #5-01-1)

13.1.1 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)

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.

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.

13.1.2 First steps in process monitoring

Starting process monitoring

> 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 in Program Run

- 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**)

> 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.
>
> **Further information:** "Overview of monitoring tasks", Page 326

- Machine further workpieces
- If applicable, assess the parts resulting from the machining operations in the **Evaluate component** area

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: "Modifying the parametrization of monitoring tasks", Page 305

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

> 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.

> If you remove a monitoring task, the previous recordings will be kept. You can add the monitoring task again later.

13.1.3 The Process Monitoring workspace (#168 / #5-01-1)

Application

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.

Requirements

- Software option Process Monitoring (#168 / #5-01-1)
- Monitoring sections have been defined with **MONITORING SECTION**
  Further information: Programming and Testing User's Manual
- Reproducible machining process in **FUNCTION MODE MILL** milling mode exists
- Program run in Full Sequence mode

Description of function

The **Process Monitoring** workspace provides information and settings for monitoring the machining process.

Areas of the Process Monitoring workspace

The **Process Monitoring** workspace is structured as follows:

- Left side: **Filter** column, which affects the table in the central area
- Center: Table with contents for setup or run-time data, can be toggled
- Right side: **Form** column with settings options or monitoring tasks, depending on the table mode

Using the **Setup table** and **Runtime table** icons in the workspace title bar, you can toggle the table mode. The contents of the **Filter** and **Form** columns depend on which table is currently active.

Table-independent area

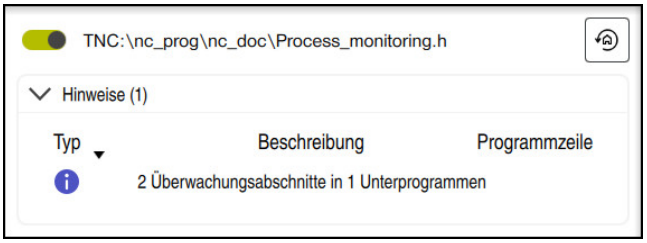


Table-independent area

Independent of the table mode, the control displays the following contents:

- Switch for activating process monitoring for this NC program
- Button for resetting the monitoring settings for this NC program to the default values.

  The control monitors the standard monitoring tasks and the default columns of the runtime table.

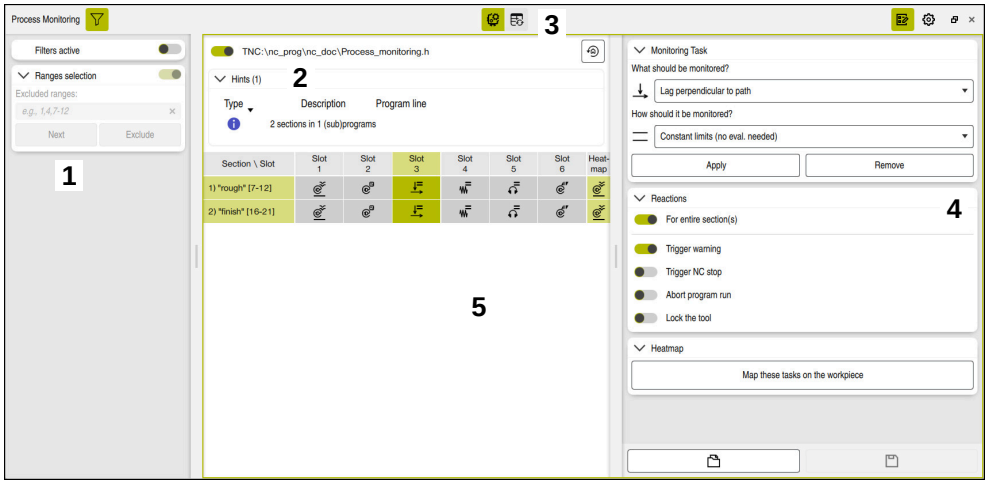
  Further information: "The Monitoring tasks area", Page 315
- Area with notes on the active NC program

  You can expand or collapse this area.

The note area provides the following information:

Column or icon	Meaning
Type	In the **Type** column, the control displays various note types.
	Information (for example, the number of monitoring sections)
	Suggestions, such as **Consider deleting all records for NC program** If you change the positioning blocks within a monitoring section, the control can no longer consider the recordings made so far. You must delete the recordings in the **Form** column. **Further information:** "Settings in the Form column", Page 316
	Warning (for example, if a monitoring section has been removed)
	Error (for example, if two monitoring sections are identical)
Description	The control displays a hint in the **Description** column.
Program line	If the information depends on an NC block number, the control displays the program name and the NC block number.

Screen layout if the setup table is active



The **Process Monitoring** workspace if the setup table is active

The **Process Monitoring** workspace contains the following items if the setup table is active:

1 **Filter** column that can be used to hide rows with monitoring sections in the table

With the **Next** and **Exclude** buttons, you can select and hide the monitoring sections.

You can activate or deactivate the filter by means of a toggle switch.

2 Table-independent area

Further information: "Table-independent area", Page 308

3 Icons for toggling the table mode between **Setup table** and **Runtime table**

4 **Form** column with program-specific settings

Further information: "Form with active settings table", Page 316

5 The Setup table

Contents of the Setup table

Each monitoring section in the NC program corresponds to one table row. The name of the row is made up as follows:

- Sequential number of the monitoring section
- Designation of the monitoring section
  - Designation as defined in **MONITORING SECTION START**
  - **SECTION** if no designation has been specified.
  - Tool name if the monitoring section starts with a tool call
- Line number of the monitoring section in square brackets

The columns contain the six possible monitoring tasks and one monitoring task whose status is represented on the workpiece in the simulation. Select this monitoring task in the **Heat map** area of the form.

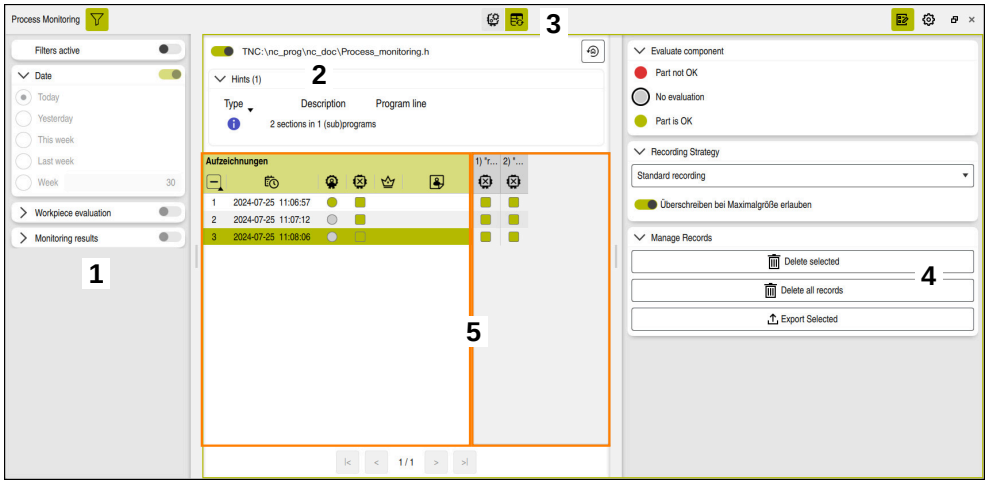
Further information: "Form with active settings table", Page 316

Using the setup table

Depending on where you tap or click on the table, you either mark the entire table, an entire row or column, or only one cell. If you select an NC block, the control will mark the associated area in the table (e.g., the row of the monitoring section).

The settings in the form reflect the current selection.

Screen layout if the runtime table is active



The **Process Monitoring** workspace if the runtime table is active

The **Process Monitoring** workspace contains the following items if the runtime table is active:

1 **Filter** column that can be used to restrict the table display to certain machining operations:
- **Date**
  Day or week of machining
- **Workpiece evaluation**
  User's assessment
- **Monitoring results**
  Process monitoring active or inactive, with or without process disturbance

You can activate or deactivate the filter by means of a toggle switch.

2 Table-independent area
Further information: "Table-independent area", Page 308

3 Icons for toggling the table mode between **Setup table** and **Runtime table**

4 **Form** column with settings or monitoring tasks, depending on the active contents of the runtime table:
- If the **Recordings** table is active, the control displays machining settings.
  Further information: "Form with active Runtime table", Page 317
- If the monitoring section tables are active, the control displays the monitoring tasks for the selected machining operation.
  Further information: "Monitoring tasks", Page 323

5 Runtime table

Contents of the runtime table

In the left area of the runtime table, the control displays the **Recordings** table with the recorded machining operations.

In the area on the right, the control displays all monitoring sections of the NC program side-by-side. You can select a monitoring section and view the results of the recorded machining operations as rows.

The control displays the first three rows of the **Recordings** table even if the area on the right is active. The following columns cannot be changed:

Icon	Meaning
	Sequential number of the recording If you activate this icon, the control will mark all rows and displays a checkmark in the icon. If you deactivate this icon, the control will remove the marks of all rows and removes the checkmark from the icon.
	Date and time at the start of the NC program
	Manual assessment of the result of the machining operation as **Part is OK** or **Part not OK** Depending on the procedure, the control uses the evaluated recordings as reference machining operations for monitoring. The control only uses the first ten good parts as reference machining operations. In the **Form** column, the control provides part evaluation, too. You can evaluate only completely executed recordings. Good parts must be representative for the machining process (for example, they must not include process interruptions or slower feed rates from proving-out).

The contents of the remaining columns can be selected in the **Configure table** window.

Further information: "The Configure table window", Page 319

If no recordings have been made yet, the control will display gray bars as placeholders in the tables.

Using the runtime table

- If you tap or click the table name, the control will display the entire table.
- If you tap or click the table column icon, the control will open the **Configure table** window.
- If you tap or click the icon in the header row, the control will sort the contents of the table by this column in ascending or descending order.
- If you tap or click a row once, it will be marked additionally. If you double-tap or double-click a row, the control will mark only this row.

Icons

The following icons are shown in the **Process Monitoring** workspace:

Icon	Meaning
	Open or close the **Filter** column
	Filters active
	Activate **Setup table**
	Activate **Runtime table**
	Open or close the **Form** column Depending on the selected item, the form has the following contents: ■ If the setup table is active, the control displays program-specific settings. **Further information:** "Form with active settings table", Page 316 ■ If the **Recordings** area in the runtime table is active, the control displays settings for the respective recording. **Further information:** "Form with active Runtime table", Page 317 ■ If the monitoring sections area is active in the runtime table, the control displays the monitoring tasks. **Further information:** "Monitoring tasks", Page 323
	Open or close the **Settings** The global settings can be accessed from the title bar of the **Process Monitoring** workspace. **Further information:** "Global settings in the Process Monitoring workspace", Page 314
	**Reset** The control monitors the standard monitoring tasks and the default columns of the runtime table.
	Colored boxes represent automatic assessments made by process monitoring.
	Colored circles are user-definable assessments.

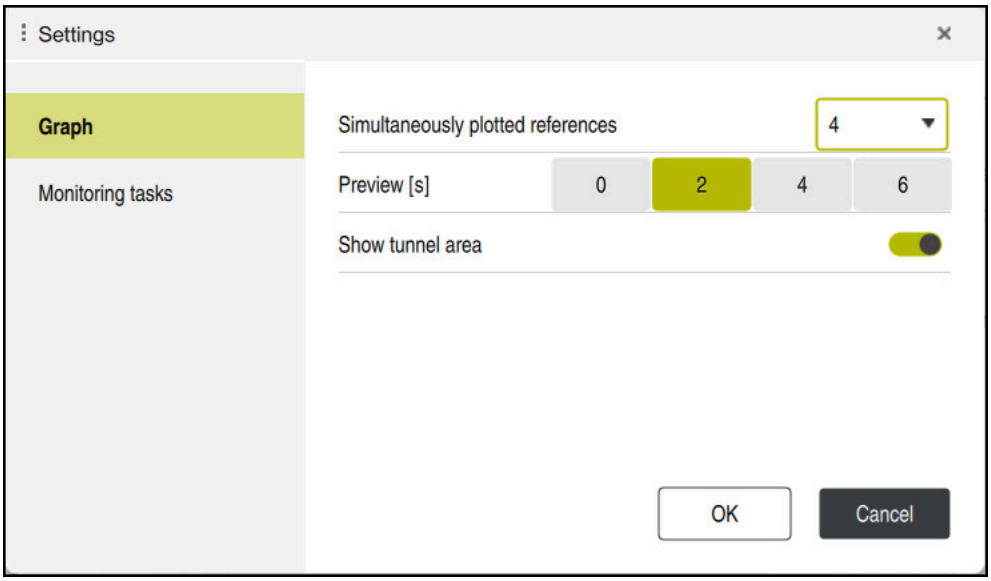
Icon	Meaning
	**Open** Load an existing monitoring template for the selected monitoring task
	**Save** Save the monitoring settings **Further information:** "Settings for parameterizing of monitoring tasks", Page 334

Further information: "Graphic display of the monitoring tasks", Page 323

Global settings in the Process Monitoring workspace

Open the global settings with an icon in the workspace title bar.

The Graph area

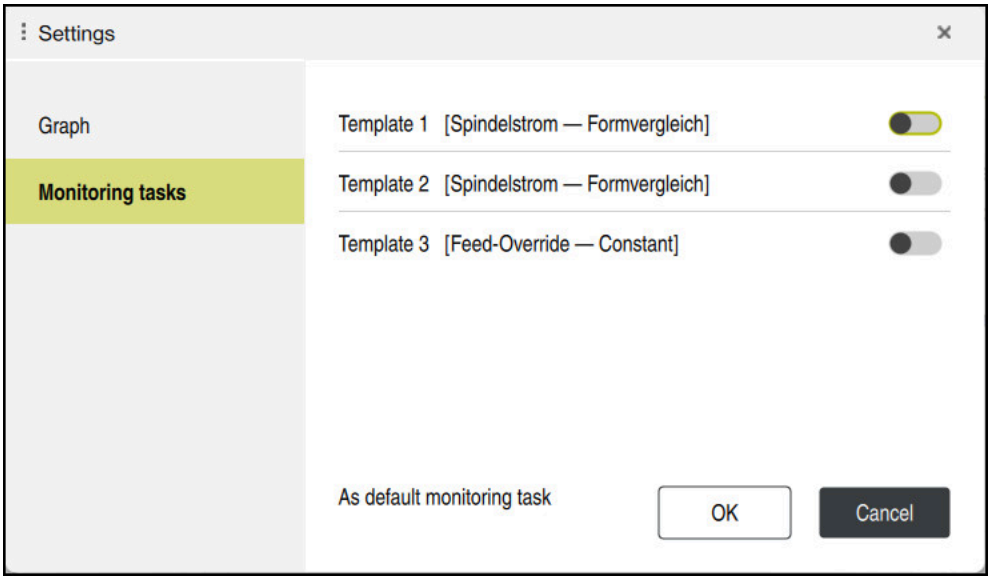


The **Graph** area of global settings

The **Graph** area offers the following settings:

Setting	Meaning
Simultaneously plotted references	Select the maximum number of recordings that the control displays simultaneously as graphs in the monitoring tasks: ■ **2** ■ **4** ■ **6** ■ **8** ■ **10**
Preview [s]	During execution, the control displays graphs of the current monitoring tasks. You can show an area for signals expected during the next seconds on the right in the graph. You can set how many seconds the control will display on the right in the graph: ■ **0** ■ **2** ■ **4** ■ **6**
Show tunnel area	When the toggle switch is active, the control displays the monitoring tunnel area in the graph on a color background. Only available for the **Tunnel** or **Constant** monitoring tasks.

The Monitoring tasks area



The **Monitoring tasks** area of the global settings

In the **Monitoring tasks** area, the control shows six saved templates for monitoring tasks in alphabetical order. This area is empty as long as you have not yet saved any templates for monitoring tasks.

You can create templates during the parameterization of monitoring tasks or in the **Form** column of the setup table. The control saves the templates in the **TNC:/system/Processmonitoring** folder.

Further information: "Settings for parameterizing of monitoring tasks", Page 334

You can save any number of templates. However, the control will only display the first six templates (in alphabetical order) in this window. If several templates refer to the same signal and procedure, the control will display only one of them. Using the icon next to the templates, you can open the templates file path in the **Files** operating mode. There, you can rename or delete templates.

The templates shown here will be used as defaults for new monitoring sections or NC programs. If you saved fewer than six unique templates, the control will additionally display templates defined by the machine manufacturer, and if there is still space left, HEIDENHAIN templates.

Settings in the Form column

Form with active settings table

When the settings table is active, the control displays the following settings in the **Form** column:

Area	Meaning
Monitoring task	For the marked cell or column, select the signal to be monitored by the control and the monitoring method. The control only displays selection options that allow a reasonable combination of signal and procedure. **Further information:** "Overview of monitoring tasks", Page 326
Reactions	You can activate or deactivate the reactions of the currently selected monitoring sections. Following a hyphen, the control shows all available reactions for one or more selected monitoring tasks. Toggle switches allow you to activate or deactivate the individual reactions. **Further information:** "Reactions", Page 333 If several monitoring tasks are marked and the statuses of the reactions do not match, the control will display the toggle switch in a center position.
Heat map	You can select one of the defined monitoring tasks whose status the control displays as a heatmap on the workpiece.
Open and **Save** buttons	You can load an already saved monitoring template for the selected monitoring task or save the current settings as a template. The control provides the same options for the parameterization of monitoring tasks. **Further information:** "Settings for parameterizing of monitoring tasks", Page 334

Further information: "Screen layout if the setup table is active", Page 309

Form with active Runtime table

If the **Recordings** area is active in the Runtime table, the control displays the following settings in the **Form** column:

Area	Meaning
Evaluate component	You can evaluate the selected machining operation as **Part is OK** or **Part not OK**. Depending on the procedure, the control uses the evaluated recordings as reference machining operations for monitoring. The control only uses the first ten good parts as reference machining operations. The control provides the same options in the third column of the Runtime table. You can evaluate only completely executed recordings. Good parts must be representative for the machining process (for example, they must not include process interruptions or slower feed rates from proving-out).
Recording strategy	■ **Record each operation completely** The control records the entire information of all machining operations. ■ **Limit: Record up to n operations** The control records machining operations up to a defined count. If the number of recorded machining operations exceeds the maximum number, the control will overwrite the last machining operation. Input: **2...999999999** ■ **Record only metainformation of operations** The control does not record any process data, but only meta-information such as the date, time and the results of monitoring tasks. The control cannot use recordings without process data as a reference machining process. This setting can be used for monitoring and logging once process monitoring has been set up completely. This setting significantly reduces the amount of data. ■ **Interval: Record each nth operation** The control does not record process data for each machining operation. You can define after which number of machining operations the control records process data. For the other machining operations, only meta-information will be recorded. Input: **2...20** ■ **Interval: Record each nth operation and critical operations** You can define after which number of machining operations the control records process data. For the other machining operations, only meta-information will be recorded. If a process interruption occurs, the control will additionally record process data for this machining operation. The control can save recordings with up to 8 GB max. for each NC program. When the **Overwrite upon maximum size** toggle switch is used, the control will overwrite the latest recordings when the maximum size is reached. If the toggle switch is inactive and the maximum memory capacity is reached, the control will interrupt process monitoring and issue an error message.

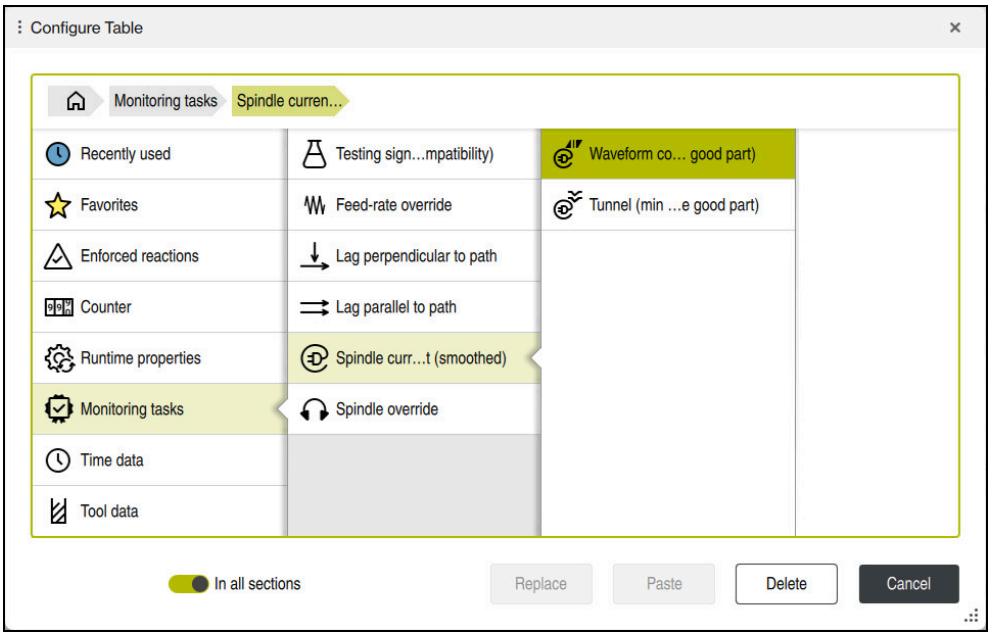
Area	Meaning
Manage recordings	■ **Delete the selection** The control deletes all selected recordings. You cannot delete the first line in this way. ■ **Delete all records** The control deletes all recordings, including the first line. ■ **Export the selection** You can save the recordings of the selected machining operations as HTML, PDF, CSV or JSON files. Refer to your machine manual. ■ The machine manufacturer defines the data to be exported by the control. ■ With the optional machine parameter **autoExportType** (no. 141602), you can define a file type that the control will automatically use to export recordings. ■ The machine manufacturer can define that the control automatically exports the recording after machining. Machine parameter **permitAutoExport** (no. 141601) defines whether the control is allowed to generate automatic recordings for the machine manufacturer.

Further information: "Screen layout if the runtime table is active", Page 310

The Configure table window

If you tap or click the icon of a column in the runtime table, the control will open the **Configure table** window. In this window, you can select the information to be displayed in the column.

Further information: "Screen layout if the runtime table is active", Page 310



The **Configure table** window

The **Configure table** window contains the following buttons:

Button	Meaning
In all monitoring sections	If you activate this toggle switch, the **Replace**, **Paste**, and **Delete** buttons will be effective for the tables of all monitoring sections. Applies to the monitoring section tables only
Replace	The control inserts the selection instead of the current column.
Paste	The control inserts the selection as a new column after the current column. Only if the **In all monitoring sections** toggle switch is active The **Recordings** table can contain up to ten columns. The tables for the monitoring sections can contain up to six columns.
Delete	The control deletes the current column. Only if the **In all monitoring sections** toggle switch is active Each table must contain at least one column.
Cancel	The control closes the **Configure table** window.

Depending on whether you selected a column in the **Recordings** area or in the monitoring sections area, the control provides different options.

The **Configure table** window can contain the following group of options:

Icon	Group	Icon	Options
	Triggered reactions		**Information reactions** Number of reactions within the NC program or monitoring section
			Warning reactions Number of reactions within the NC program or monitoring section
			Stop reactions Number of reactions within the NC program or monitoring section
			Most critical reaction Icon of the most critical reaction **Further information:** "Reactions", Page 333
	Counter		**Counter current value (start)** At the beginning of the NC program or monitoring section
			Counter target value (start) At the beginning of the NC program or monitoring section
			Counter current value (stop) At the end of the NC program or monitoring section
			Counter target value (stop) At the end of the NC program or monitoring section
	Workpiece evaluation		**Manual evaluation**
	Runtime properties		**Resulting value** Result of the entire machining operation You can select the display mode. **Further information:** "Display mode", Page 322
			Note You can enter a comment.
			Contains runtime data Check box Only recordings that contain run-time data can be selected as reference machining operations.
			Pallet name Only if the NC program is part of pallet machining or pallet handling
	Monitoring tasks		You can select the monitoring task and the display mode. **Further information:** "Monitoring tasks", Page 323 **Further information:** "Display mode", Page 322

Icon	Group	Icon	Options
	Time data		**Date**
			Time Time at the beginning of the NC program or monitoring section
			Date and time Date and time at the beginning of the NC program or monitoring section
			Duration Duration of the NC program or monitoring section in seconds
	Tool data		**Further information:** "Parameters of the tool table tool.t", Page 457

Colored boxes represent automatic assessments made by process monitoring. Colored circles are user-definable assessments.

Display mode

For monitoring tasks and the **Resulting value**, you can specify how the control will display the result:

Display mode	Icon	Options
Worst or **Best**		**Best** The control shows the smallest deviation from the monitoring tasks during machining (i.e., the best result). Using this option, you can see, for example, whether there was a process disturbance before or during a monitoring section.
		**Worst** The control shows the greatest deviation from the monitoring tasks during machining (i.e., the worst result). Using this option, you can see at a glance whether there was a process disturbance during machining.
Resulting quantity (discrete) or **Resulting quantity (color gradient)**	No icon	**Resulting quantity (color gradient)** The control displays the result using a color scale from green to dark red. With this option, you can, for example, see whether a machining operation was approaching the error limit.
	No icon	**Resulting quantity (discrete)** The control uses the following colors for the results display: ■ Green: Machining was in the range of the lower half of the tolerance ■ Yellow: In the range of the upper half of the tolerance, but below the error limit ■ Red: Error limit exceeded Using this option, you can see quickly whether error limits were exceeded.

If no result is available yet, the control displays a gray icon.

13.1.4 Monitoring tasks

A monitoring task consists of the following properties:

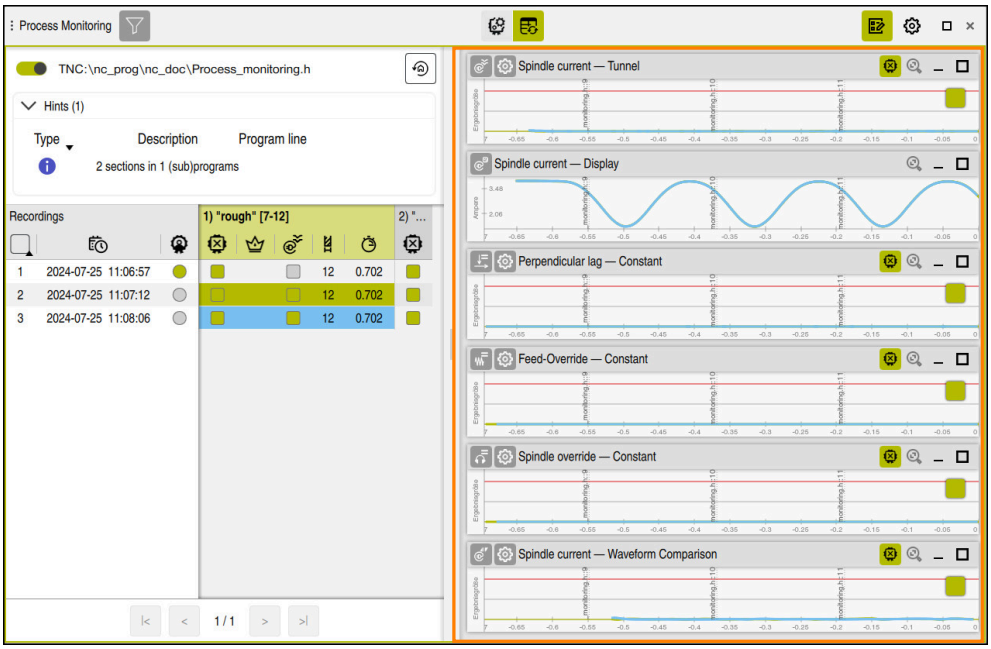
- Signal (e.g., spindle current)
- Procedure for evaluating the signal (e.g., waveform comparison)
- One or more parameters (e.g., tolerance), depending on the selected procedure
- Reactions (e.g., stopping the NC program)

The control contains pre-defined monitoring tasks.

Refer to your machine manual.
The following monitoring tasks are included in the standard scope and have been configured by HEIDENHAIN. The machine manufacturer cannot modify these monitoring tasks, but can define further monitoring tasks.

For each monitoring section, you can define up to six monitoring tasks. The control displays the monitoring tasks in the **Form** column as graphs.

Graphic display of the monitoring tasks



The **Form** column with graphs of the monitoring tasks

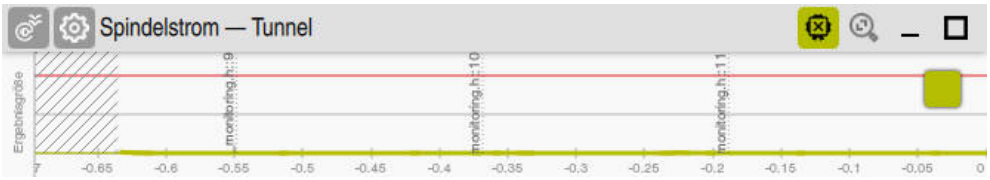
If the tables of the monitoring tasks are active in the run-time table, the control displays the graphs of the monitoring tasks in the **Form** column. The control displays the values of the selected recordings in the graphs.

During program run, the control always displays the momentary values for the monitoring tasks.

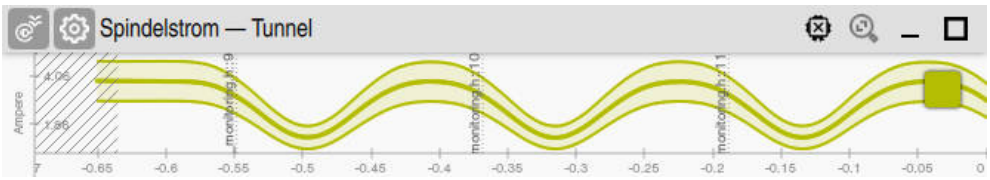
Each graph represents the machining operation as a resulting value or signal curve. The signal curve additionally shows the reference machining operations used as well as a vertical axis with the relevant unit. The time axis is divided into seconds, or into minutes for longer monitoring sections.

If the control cannot evaluate a monitoring task, it displays the graph with gray hatch marks. If you click or tap this gray hatched area, the control opens a window. In this window the control shows reasons for why the monitoring task was not evaluated.

Icons of monitoring tasks



Graph as resulting value relative to the error limit



Graph as non-evaluated signal curve with tunnel

The graphs of the monitoring tasks contain the following icons:

Icon	Meaning
	The control displays the icon for the monitoring task. The icon is a combination of the signal and the procedure. If you select the icon, the control opens a selection window where you can modify or remove the monitoring task. **Further information:** "Overview of monitoring tasks", Page 326
	Open or close the **Settings** for parameterization **Further information:** "Settings for parameterizing of monitoring tasks", Page 334
	Change **Signal representation** You can change between the following signal representations: ■ Resulting value The resulting value shows the evaluated signal relative to the error limits. When the signal approaches the red line, machining deviates from the recordings. If the current machining process exceeds the red line, the monitoring task triggers the defined reactions (e.g., NC stop). ■ Signal curve The signal curve shows the non-evaluated signal as an absolute value. If the selected procedure uses a tunnel, the control displays the tunnel around the signal by means of broken lines. Depending on the settings, the control displays the tunnel with a color background. If the signal goes outside the tunnel for the defined holding time, the monitoring task triggers the defined reactions.
	Reset scaling Show graph of the entire monitoring section If the icon is dimmed, the control displays the entire graph.

Icon	Meaning
▢	Rectangular color icons are automatic assessments by process monitoring. The control displays the worst resulting value of the entire monitoring section, regardless of the visible area of the graph.

Using the graphs

- You can zoom in or out of the graph horizontally by scrolling or dragging.
- You can shift the graph by swiping or while pulling with the left mouse button pressed.
- The control marks the selected NC block number within the graph by a vertical green line.
- If you double-click or double-tap a position within the graph, the control selects the corresponding NC block in the NC program and in the graph.

Overview of monitoring tasks

The table below contains an overview of the monitoring tasks. Detailed information about the following properties can be found in the content below:

- Procedure
  Further information: "Procedure", Page 329
- Reactions
  Further information: "Reactions", Page 333

The first six monitoring tasks are the standard HEIDENHAIN monitoring tasks. If you or the machine manufacturer have not defined any templates, these monitoring tasks are active by default in a new NC program or monitoring section. You can also modify the monitoring tasks.

Further information: "Changing the monitoring task", Page 305

The control provides the following monitoring tasks:

Icon	Meaning	
	Spindle current — Tunnel	
	Sample cases:	■ Identifying broken tools ■ Identifying a missing tool ■ Identifying faulty clamping ■ Identifying missing pre-machining.
	Signal:	Spindle current (smoothed, without spindle acceleration)
	Procedure:	Tunnel
	Requirement:	At least one good part
	Parameters:	■ Tolerance percentage of mean value of reference signals in % ■ Dynamic tunnel width: Multiple of measured standard deviation σ of the reference signals ■ Static tunnel width in A ■ Hold time for reactions in ms
	Spindle current — Display	
	Sample case:	Pure display without monitoring
	Signal:	Spindle current (smoothed)
	Procedure:	Graph display
	Requirement:	No assessment required
	Perpendicular lag — Constant	
	Sample case:	Identifying contouring deviations vertically with respect to the contour run
	Signal:	Lag of all axes vertically with respect to contour run
	Procedure:	Constant Fixed limits that are independent of the signal
	Requirement:	No assessment required
	Parameters:	■ Upper limit for lag in µm ■ Hold time for reactions in ms

Icon	Meaning	
	Feed-Override — Constant	
	Sample case:	Identifying feed rate override deviations
	Signal:	Feed rate override
	Procedure:	Constant Fixed limits that are independent of the signal
	Requirement:	No assessment required
	Parameters:	■ Upper limit for the override in % ■ Lower limit for the override in % ■ Hold time for reactions in ms
	Spindle override — Constant	
	Sample case:	Identifying changes of spindle override
	Signal:	Spindle override
	Procedure:	Constant Fixed limits that are independent of the signal
	Requirement:	No assessment required
	Parameters:	■ Upper limit for the override in % ■ Lower limit for the override in % ■ Hold time for reactions in ms
	Spindle current — Waveform Comparison	
	Sample cases:	■ Identifying broken tools ■ Identifying a missing tool ■ Identifying faulty clamping ■ Identifying missing pre-machining.
	Signal:	Spindle current (without spindle acceleration)
	Procedure:	Waveform comparison
	Requirement:	At least one good part
	Parameters:	Tolerance of waveform with the reference signals
	Feed per tooth — Display	
	Sample case:	Pure display without monitoring
	Signal:	Tooth feed rate **FZ**
	Procedure:	Graph display
	Requirement:	Number of teeth **CUT** of the tool defined in tool management

Icon	Meaning
	**Perpendicular servo lag — Tunnel** **Sample case:** Identifies path deviations perpendicular to the path direction (e.g., mapping the size tolerance) **Signal:** Lag of all axes vertically with respect to contour run **Procedure:** Tunnel **Requirement:** At least one good part **Parameters:** ■ Tolerance percentage of mean value of reference signals in % ■ Dynamic tunnel width: Multiple of measured standard deviation σ of the reference signals ■ Static tunnel width in A ■ Hold time for reactions in ms
	**Parallel lag — Constant** **Sample case:** Identifying contouring deviations in parallel with the contour run **Signal:** Lag of all axes in parallel with contour run **Procedure:** Constant Fixed limits that are independent of the signal **Requirement:** No assessment required **Parameters:** ■ Upper limit for lag in µm ■ Hold time for reactions in ms
	**Parallel servo lag — Tunnel** **Sample case:** Identifying contouring deviations in parallel with the contour run **Signal:** Lag of all axes in parallel with contour run **Procedure:** Tunnel **Requirement:** At least one good part **Parameters:** ■ Tolerance percentage of mean value of reference signals in % ■ Dynamic tunnel width: Multiple of measured standard deviation σ of the reference signals ■ Static tunnel width in A ■ Hold time for reactions in ms
	**Testing signal — Display**  This monitoring task is intended for test purposes and should be used only if requested by HEIDENHAIN or by the machine manufacturer! **Sample cases:** Pure display without monitoring **Signal:** Process signal The signal may change between different software statuses. Compatibility between software updates is not guaranteed. **Procedure:** Graph display **Requirement:** No assessment required

Icon	Meaning
	Testing signal — Waveform Comparison

This monitoring task is intended for test purposes and should be used only if requested by HEIDENHAIN or by the machine manufacturer!

Sample cases:
- Identifying broken tools
- Identifying a missing tool
- Identifying faulty clamping
- Identifying missing pre-machining.

Signal: Process signal

The signal may change between different software statuses. Compatibility between software updates is not guaranteed.

Procedure: Waveform comparison

Requirement: At least one good part

Parameters: Tolerance of waveform with the reference signals

Testing signal — Tunnel

This monitoring task is intended for test purposes and should be used only if requested by HEIDENHAIN or by the machine manufacturer!

Sample cases:
- Identifying broken tools
- Identifying a missing tool
- Identifying faulty clamping
- Identifying missing pre-machining.

Signal: Process signal

The signal may change between different software statuses. Compatibility between software updates is not guaranteed.

Procedure: Tunnel

Requirement: At least one good part

Parameters:
- Tolerance percentage of mean value of reference signals in %
- Dynamic tunnel width: Multiple of measured standard deviation σ of the reference signals
- Static tunnel width in A
- Hold time for reactions in ms

Procedure

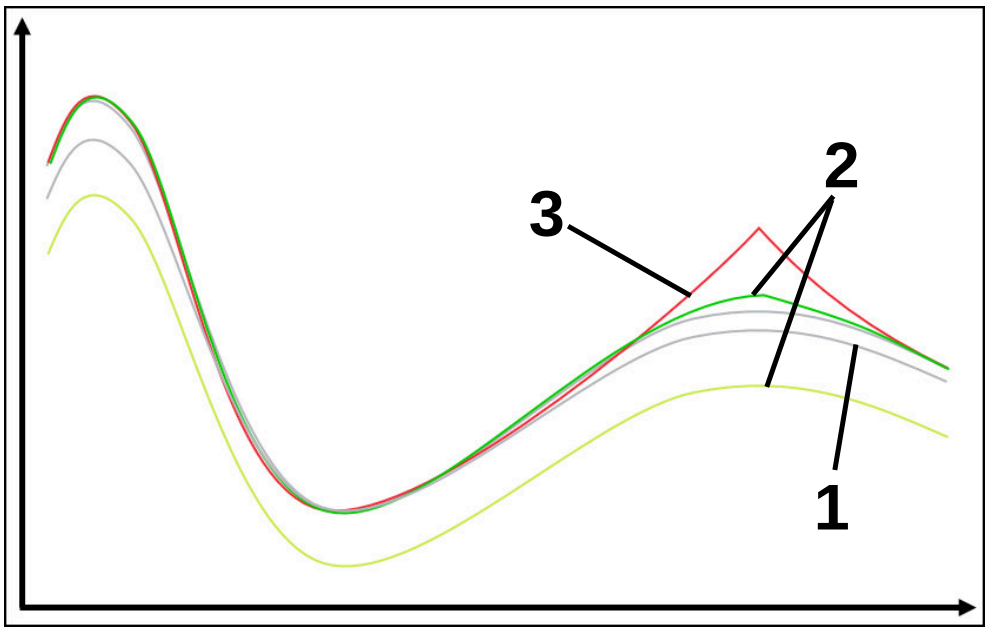
Process monitoring offers the following procedures:

- Waveform comparison
  Further information: "Waveform comparison", Page 330
- Tunnel
  Further information: "Tunnel", Page 331
- Display
  Further information: "Display", Page 332
- Constant
  Further information: "Constant", Page 332

Waveform comparison

In the **Waveform comparison** procedure, the control compares the current signal wave with the recordings of good parts at short time intervals. If the wave deviates too strongly, the monitoring task identifies a potential fault. A long-term signal drift will not modify the waveform and will therefore not cause any reaction.

In this procedure, the control will not display any error limits in the signal run.



1 These recordings are assessed as good parts and are used as reference machining operations.

2 Machining with a slight deviation

For one machining operation, the signal waveform deviates slightly from the reference machining operations. In the other machining operation, the signal strength deviates from the reference machining operations, but the waveform is identical. These machining operations do not trigger a reaction.

3 Machining with a heavy deviation

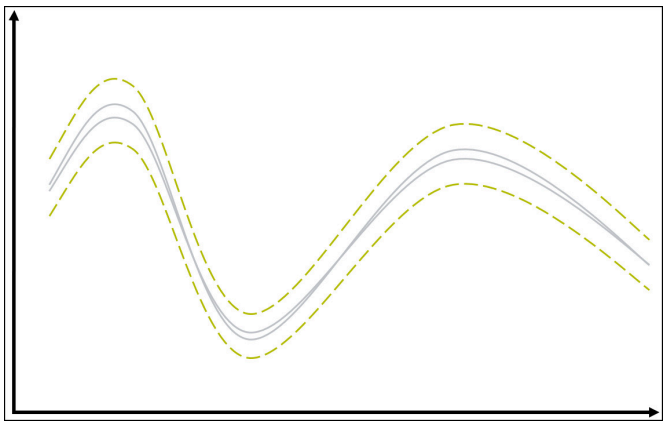
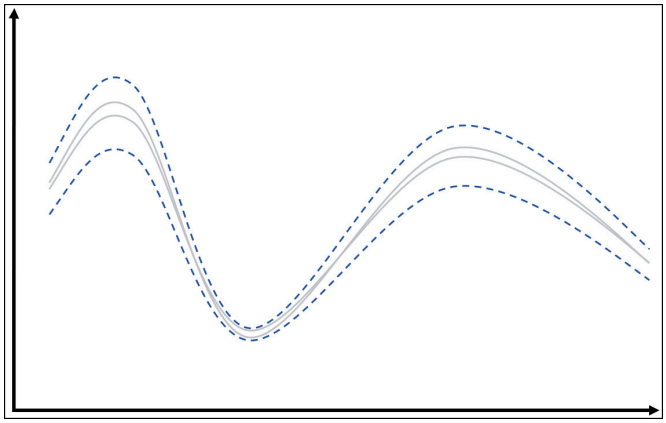
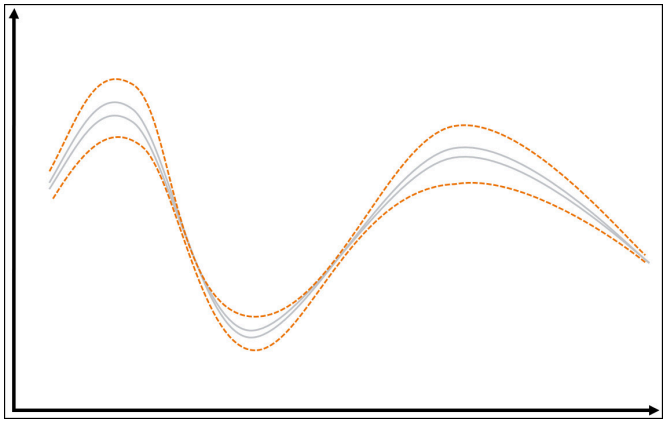
The waveform of this machining operation deviates heavily from the previous records and will trigger the configured reactions.

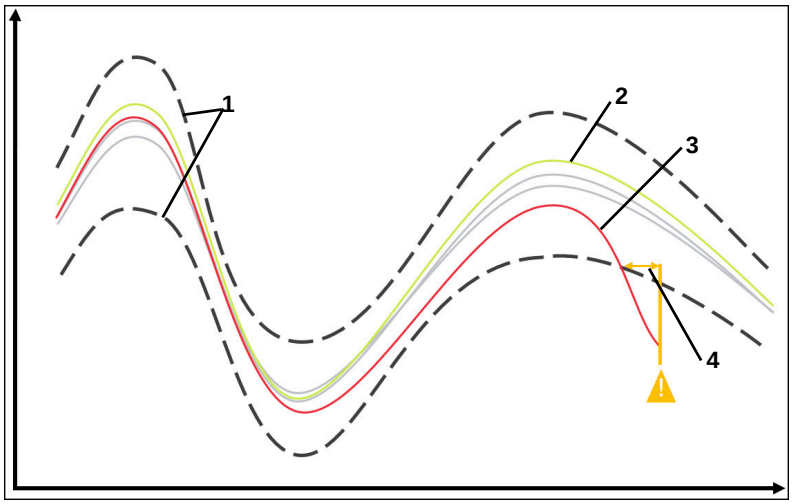
Tunnel

Using the **Tunnel** procedure, the control monitors whether the parts produced by the current machining operation are within the range of the previously selected good parts, including the tunnel width.

This method reacts to both short-term changes and long-term signal drifts. A short-term change may be due to tool breakage, for example. A long-term drift may originate from a change in temperature, for example.

The tunnel width limits can be defined by the following values:

Value	Graph
Absolute, static tolerance (e.g., tunnel width 30 A) The gray lines represent the reference machining operations	
Percentage deviation from the reference machining operations (e.g., 30 %)	
Multiple of the standard deviation σ of the reference machining operations (e.g., 3 x σ)	



1 — Error limits

The error limits result from the total of all three possible values.

2 — Machining with a slight deviation

This machining operation deviates slightly from the previous recordings, but is still within the error limits.

3 — Machining with a significant deviation

This machining operation deviates considerably from the previous recordings. The machining operation exceeds the error limit and triggers the configured reactions after the defined hold time has elapsed.

4 — Hold time

If the value is below or above the error limits, the control triggers the configured reactions after the defined hold time has elapsed.

Display

In the **Display** procedure, the control displays the run of the selected signal of current machining. The control does not carry out any reactions, you can only check the recording visually.

Constant

In the **Constant** procedure, the control monitors if the current machining is within the defined error limits. The error limits result from the defined tolerances which are independent of the signal. This makes the monitoring task perform monitoring with this procedure starting from the first machining operation, and does not require any assessments of recordings.

Reactions

Refer to your machine manual.
The machine manufacturer can define further reactions.

If a signal exceeds the error limits for longer than the defined hold time, the monitoring task can execute one or more reactions.

You can define the reactions in the following locations of the **Process Monitoring** workspace:

- **Form** column if the setup table is active
  Further information: "Form with active settings table", Page 316
- Settings for the parameterization of each monitoring task
  Further information: "Settings for parameterizing of monitoring tasks", Page 334

You can choose from the following reactions, depending on the monitoring task:

Reaction	Meaning
Trigger warning	The control displays a warning in the notification menu. **Further information:** "Message menu on the information bar", Page 365
Trigger NC stop	The control stops the NC program. You can then check the machining status. If you find that there is no serious error, you can resume the NC program. The control reactivates process monitoring in the next monitoring section.
Abort program run	The control stops the NC program. In this case, the NC program cannot be resumed. The machine manufacturer can define the behavior of the control in connection with pallet machining in case a program is aborted (e.g., continue machining the workpieces on the next pallet).
Lock the tool	The control blocks the tool in the tool management. **Further information:** "Tool management ", Page 197

Settings for parameterizing of monitoring tasks

Use the **Settings** icon in a monitoring task to change its parameterization.

Parameterization of a monitoring task

When selecting the settings of a monitoring task, the control displays two areas:

1 Parameterization of selected recording

The control dims the parameterization that was active at the time of the selected recording.

2 Preview of current parameterization

The control displays the current parameterization for the monitoring task. When changing the settings, the control displays which effects the changes have on the selected machining operation.

The control uses a colored box icon to highlight the worst resulting value in the visible range. To see the worst resulting value of the entire monitoring section, display the entire graph.

The settings of monitoring tasks contain the icons and buttons below:

Icon or button	Meaning
Apply	Restore values from the left view
Cancel	Reject parameterization changes
OK	Save changes to the parameterization
	**Open** You can load an existing monitoring template for the selected monitoring task.

Icon or button	Meaning
	**Save** You can save the parameterization of the current monitoring task as a template. You can use the monitoring templates for other monitoring sections or in other NC programs as well. The control saves the template under a user-defined name in the **TNC:/system/Processmonitoring** folder. The control displays six templates in alphabetical order in the global settings. These six templates will be used for new monitoring sections or NC programs. **Further information:** "Global settings in the Process Monitoring workspace", Page 314

Basics of operation

- You can also use the left and right arrow keys to change the setting of the slider.
- For parameterization, you can select the **Signal display** icon to toggle between the unevaluated signal run and the resulting value, relative to the error limits.
- It may be necessary to adjust the parameterization to the machining operation (e.g., when decreasing the tunnel width for finishing operations).
- With the **Perpendicular servo lag — Tunnel** monitoring task and an absolute, static tolerance, you can check a size tolerance compared to reference machining.

14

Opening CAD files with CAD Viewer

14.1 Fundamentals

Application

CAD Viewer supports the following standard file types that can be opened directly in the control:

File type	Extension	Format
STEP	*.stp and *.step	■ AP 203 ■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015 ■ ASCII
STL	*.stl	■ Binary ■ ASCII

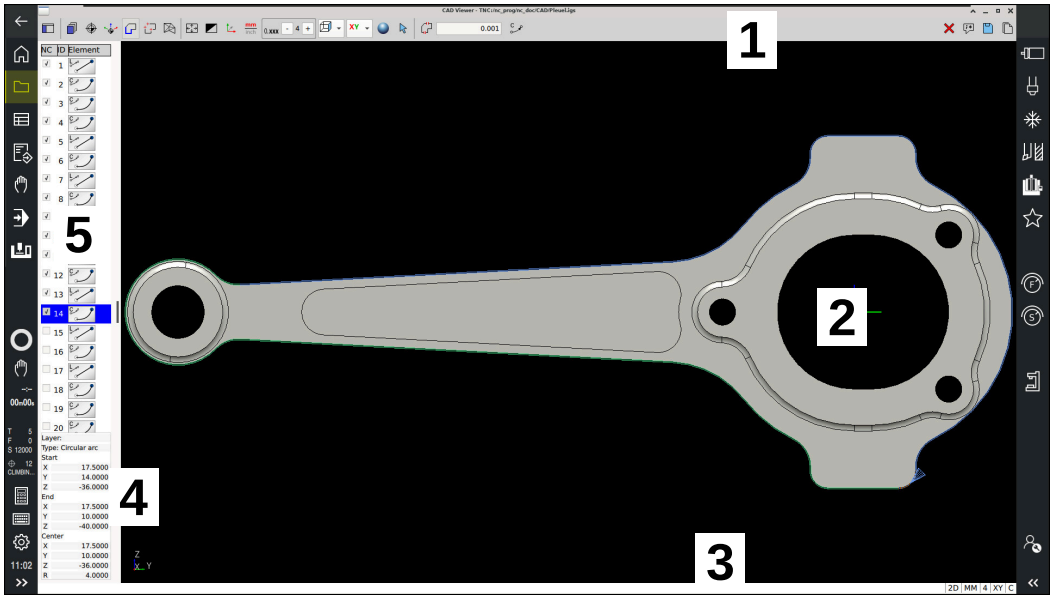
CAD Viewer runs as a separate application on the third desktop of the control.

Related topics

- Creating 2D sketches on the control
  Further information: Programming and Testing User's Manual

Description of function

Screen layout



CAD file open in **CAD Viewer**

CAD Viewer consists of the following areas:

1 Menu bar
Further information: "Menu bar icons", Page 340

2 Graphics area
The CAD model is displayed in the graphics window.

3 Status bar
The status bar contains the active settings.

4 Element information area
Further information: "Element Information area", Page 342

5 List View area
The List View area displays information on the active function (e.g., available layers or the position of the workpiece preset).

Menu bar icons

The menu bar contains the following icons:

Icon	Meaning
	**Show sidebar** Show, enlarge, or hide the List View area
	**Display the layer** Display the layer(s) in the List View area **Further information:** "Layer", Page 342
  	**Preset** Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD file", Page 343
 	**Datum** Set the datum Datum has been set **Further information:** "Workpiece datum in the CAD file", Page 346
	**Contour** Select contour (#42 / #1-03-1) **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348
	**Positions** Select positions (#42 / #1-03-1) **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348
	**3D mesh** Create a 3D mesh (#152 / #1-04-1) **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 356
	**Show all** Set the zoom to the largest possible view of the complete graphics
	**Inverted colors** Change the background color (black or white)
	Toggle between 2D and 3D modes
	Set the unit of measure (mm or inches) **CAD Viewer** performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts all values to inches. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348

Icon	Meaning
0.01 0.001	**Number of decimal places** Select decimal places: **3...7** for mm and **4...8** for inches The decimal places define the resolution and the number of segments during linearization. **Further information:** "Applying contours", Page 349 Default setting: 4 decimal places with **mm**, and 5 decimal places with **inch** as the unit of measure
	**Set perspective** Switch between various views of the model (e.g., **Top**)
XY	**Axes** Select the working plane: ■ **XY** ■ **YZ** ■ **ZX** ■ **ZXØ** In the **ZXØ** working plane, you can select turning contours (#50 / #4-03-1). If you take over a contour or position, the control outputs the NC program in the selected working plane. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348
	Toggle a 3D model between a solid model and a wire-frame model.
	**Rotate** the CAD model
	**Select** of contour elements (e.g., to obtain element information) **Further information:** "Element Information area", Page 342
	**Add** contour elements
	Remove contour elements The icon shows the current mode. Clicking the icon activates the next mode. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 348
	Undo
	**Delete entire list**
	**Setting for whether comments are written to NC output files.** Adding comments with workpiece information to the NC program (#42 / #1-03-1) The control displays the icon if you select the **Contour** or **Positions** mode. **Further information:** "Selecting and saving a contour", Page 351

Icon	Meaning
	**Save entire list content to a file**
	**Copy entire list contents to clipboard** The control retains the content of the clipboard only as long as **CAD Viewer** is open.

Element Information area

In the Element Information area, the following information is displayed for the selected element of the CAD file:

- Associated layer
- Element type
- Point type:
  - Point coordinates
- Line type:
  - Coordinates of the starting point
  - Coordinates of the end point
- Circular arc or circle type:
  - Coordinates of the starting point
  - Coordinates of the end point
  - Coordinates of the center point
  - Radius

The control always shows the **X**, **Y**, and **Z** coordinates. In 2D mode, the Z coordinate is dimmed.

Layer

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.

If the name of the layer is not shown completely in the List View area, you can use the **Show sidebar** icon to enlarge this area.

Use the **Display the layer** icon to display all the layers of the file in the List View area. Use the check box in front of the name to show and hide individual layers.

When you open a CAD file in **CAD Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

  Further information: Programming and Testing User's Manual
- When you select a layer in the List View area, you can press the spacebar to show and hide the layer.
- **CAD Viewer** allows you to open CAD files consisting of any number of triangles.

14.2 Workpiece preset in the CAD file

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

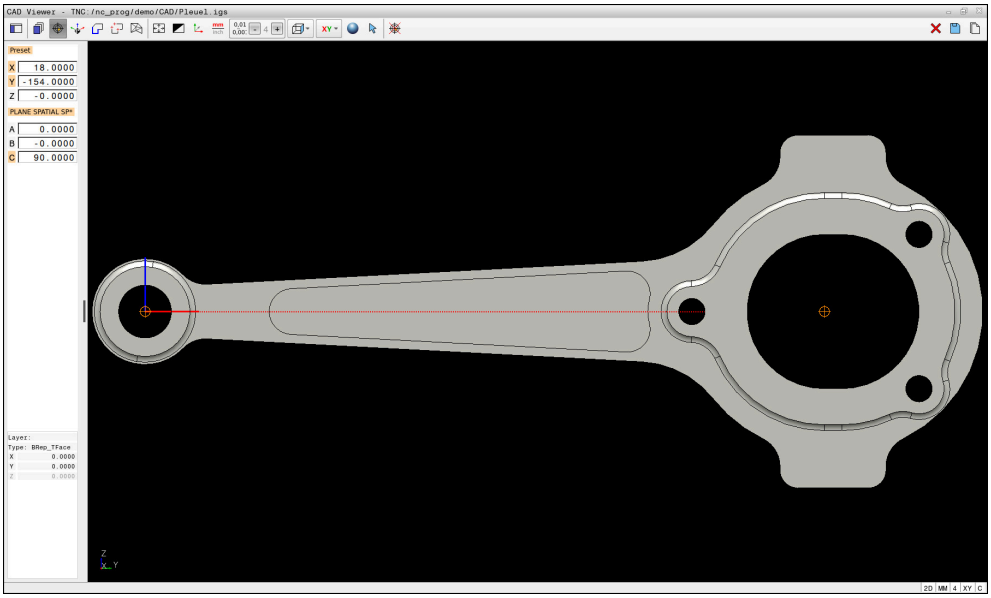
  Further information: "Presets in the machine", Page 174

Description of function

When you select the **Preset** icon, the control displays the following information in the list view area:

- Distance between the defined preset and the drawing datum
- Orientation of the working plane

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD file

You can position the preset at the following locations:

- By direct numerical input in the List View area
- For lines:
  - Starting point
  - Midpoint
  - End point
- For circular arcs:
  - Starting point
  - Midpoint
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center

- At the intersection between:
  - Two lines, even if the point of intersection is actually on the extension of one of the lines
  - Line and circular arc
  - Line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option (#42 / #1-03-1).

The control retains the content of the clipboard only as long as **CAD Viewer** is open.

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

14.2.1 Setting the workpiece preset or workpiece datum and orienting the coordinate system

- The following instructions apply when using a mouse. You can also perform these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 89
- The following instructions also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:

- ▶ Select **Preset**

- ▶ Select **Rotate**, if necessary

- > The icon changes, and the control activates the **Select** mode.
- ▶ Position the cursor on the desired element
- > If you are using a mouse, the control displays selectable presets for the element using gray icons.
- ▶ Click the icon at the desired position
- > The control sets the workpiece preset to the selected position. The control turns the icon green.
- ▶ Orient the working plane, if required

Setting the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at the intersection of lines, full circles, and arcs.

To set the workpiece preset at the intersection of two elements:

- Select **Preset**

- Select **Rotate**, if necessary

> The icon changes, and the control activates the **Select** mode.

- Click on the first element
> The control highlights the element in color.
- Click on the second element
> The control sets the workpiece preset at the point of intersection of the two elements. The control marks the workpiece preset with a green symbol.
- Orient the working plane, if required

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Orienting the working plane

The following requirements must be met in order to orient the working plane:

- Preset has been defined
- There are elements next to the preset that can be used for the desired orientation

To orient the working plane:

- Select an element in the positive direction of the X axis
> The control orients the X axis.
> The control changes the **C** angle in the List View area.
- Select an element in the positive direction of the Y axis
> The control orients the Y and Z axes.
> The control changes the **A** and **C** angles in the List View area.

14.3 Workpiece datum in the CAD file

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a working plane.

Related topics

- Presets in the machine

  Further information: "Presets in the machine", Page 174

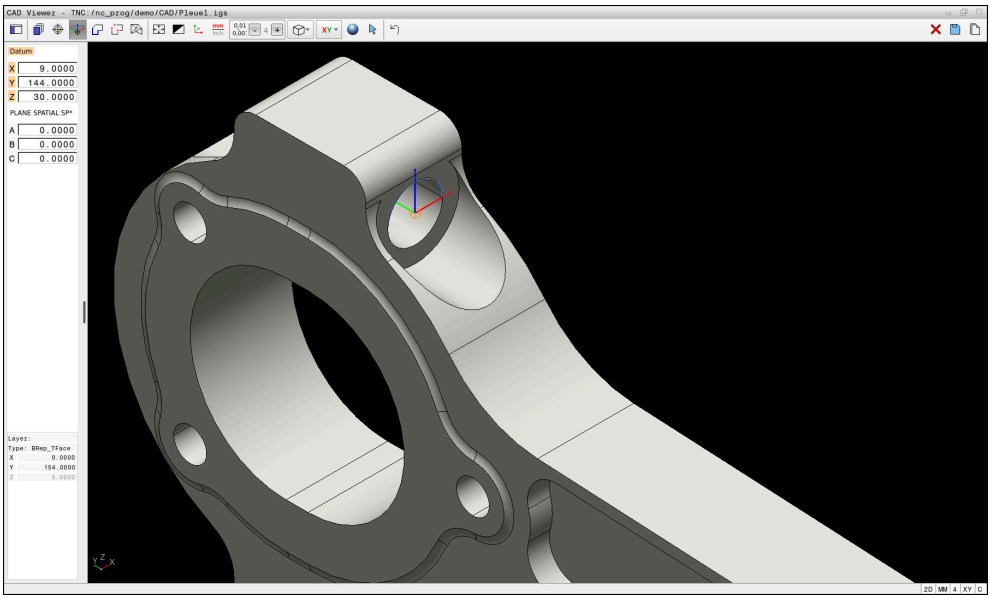
Description of function

When you select the **Datum** icon, the control displays the following information in the list view area:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the working plane

You can apply a workpiece datum set in CAD Viewer and shift it, if required, by entering values directly in the List View area.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the working plane can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD file", Page 343

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Setting the workpiece preset or workpiece datum and orienting the coordinate system", Page 344

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option (#42 / #1-03-1).

The control retains the content of the clipboard only as long as **CAD Viewer** is open.

14.4 Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)

Application

You can open CAD files directly on the control to extract contours or machining positions from them. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs by default contain only **L** and **CC**/**C** blocks.

Related topics

- Using point tables
  Further information: User's Manual for Machining Cycles

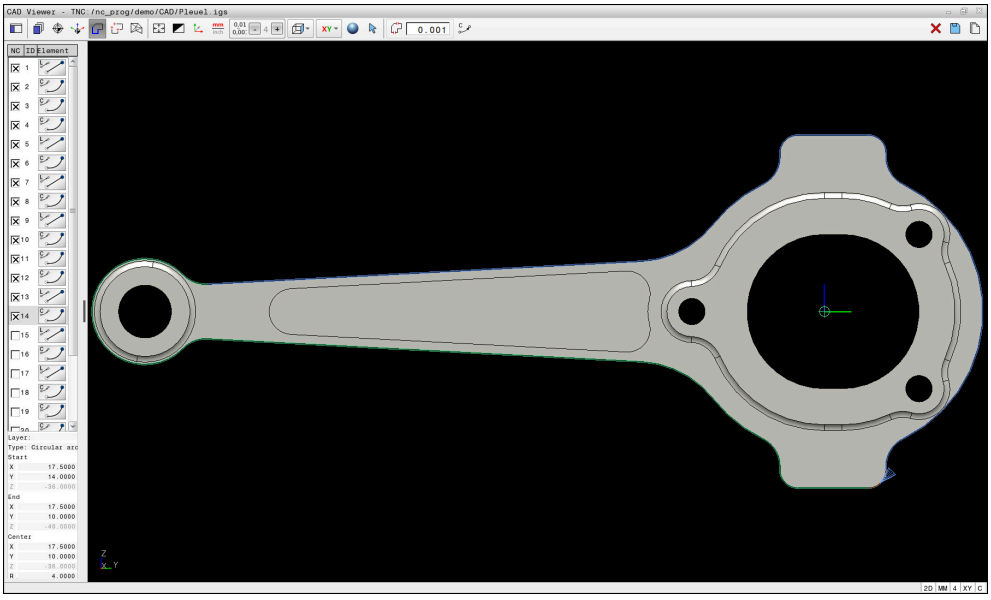
Requirement

- Software option CAD Import (#42 / #1-03-1)

Description of function

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to additional software tools (e.g., **Leafpad** or **Gnumeric**).

Further information: "Opening files with additional software", Page 679



CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Meaning
	**Set the transition tolerance** The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred during drawing creation. The default setting is 0.001 mm.
 	**C** or **CR** You can select whether the control will output circular contours **C** or **CR** in the NC program.
	**Show connections between two positions** The control hides and displays the tool paths between the positions.
	**Apply path optimization** The control optimizes the tool traverse movement between the machining positions. When you select the icon again, the control will discard the optimization.
	**Find circles according to diameter range. Load center coordinates to the position list** The control opens the **Find circle centers by diameter range** window. You can filter by diameters as well as by depths.

Applying contours

The following elements can be selected as a contour:

- Line segment
- Full circle
- Pitch circle
- Polyline
- Any curves (e.g., splines, ellipses)

Linearization

CAD Viewer linearizes all of the contours that are not in the working plane.

During linearization, **CAD Viewer** subdivides a contour into individual segments. From these segments, CAD Import creates straight lines **L** and circular arcs **C** or **CR** that are as long as possible.

Thanks to linearization, it is also possible to import contours with CAD Import that cannot be programmed with the path functions of the control, such as splines.

The higher you define the resolution by specifying decimal places, the lower is the deviation from the imported contour. In any case, the deviation is less than 0.001 mm or 0.0001 inches.

Further information: "Screen layout", Page 339

> You can prevent the linearization of, for example, circles that are not in the working plane. Select the working plane in which the circle has been defined.

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

Using CAD Import, you can also import contours for turning (#50 / #4-03-1). Before selecting a turning contour, you must set the preset on the rotary axis. CAD Import saves turning contours with Z and X coordinates and outputs the X coordinates as diameter values. Any contour elements below the rotary axis cannot be selected and are highlighted in gray.

Applying positions

You can also use the CAD Import to save positions (e.g., for holes).

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection
- Multiple selection within a range
- Multiple selection using search filters

Further information: "Selecting positions", Page 353

The following file types are available:

- Point table (**.PNT**)
- Klartext program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

CAD Viewer also considers circles that consist of two semicircles to be one machining position.

Multi-selection filter settings

If you use the quick-selection function to mark positions, the **Find circle centers by diameter range** window opens. You can filter the diameter or depth values, referencing the workpiece datum, by means of the buttons below the displayed value. The control will only load the selected diameter or depth values.

The **Find circle centers by diameter range** window provides the following buttons:

Button	Meaning
| <<	■ The control shows the smallest diameter found. ■ The control shows the smallest depth found. This filter is active by default.
<<	■ The control sets the filter for the largest diameter to the value selected for the smallest diameter. ■ The control sets the filter for the largest depth to the value selected for the smallest depth.
<	■ The control shows the next smaller diameter found. ■ The control shows the next smaller depth found.
>	■ The control shows the next larger diameter found. ■ The control shows the next larger depth found.
>>	■ The control sets the filter for the smallest diameter to the value selected for the largest diameter. ■ The control sets the filter for the smallest depth to the value selected for the largest depth.
>> |	■ The control shows the largest diameter found. ■ The control shows the largest depth found. This filter is active by default.

14.4.1 Selecting and saving a contour

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 89
- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.

Selecting a contour with existing contour elements

To select and save a contour with existing contour elements:

- Select **Contour**
- Place the cursor on the first contour element
- The control shows the suggested direction of rotation as a dashed line.
- If necessary, move the cursor towards the more distant end point.
- The control changes the suggested direction of rotation.
- Select the contour element
- The selected contour element is displayed in blue and is marked in the List View area.
- Other contour elements are shown in green.

> The control suggests the contour that deviates least from the suggested direction. To change the suggested contour path, you can select paths independently of the existing contour elements

- Select the last desired contour element
- All contour elements up to the selected element are shown in blue and are marked in the List View area.

- Activate the output of comments with workpiece information, if desired

- Select **Save entire list content to a file**
- The control opens the **Define file name for contour program** window.
- Enter the desired name
- Select the path to the storage location
- Select **Save**
- The selected contour is saved as an NC program.

- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you select an element with the CTRL key pressed, it is deselected for export.

Selecting paths independent of existing contour elements

To select a path independent of existing contour elements:

- Select **Contour**

- Select **Select**, if necessary
- > The icon changes, and the control activates the **Add** mode.
- Place the cursor relative to the desired contour element
- > The control displays selectable points:
  - End point or center point of a line or curve
  - Quadrant transitions or center of a circle
  - Points of intersection between existing elements
- Select the desired point
- Select more contour elements

If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line.
If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

Saving a contour as a workpiece blank definition (#50 / #4-03-1)

For a workpiece blank definition in turning mode, a closed contour is required.

NOTICE

Danger of collision!

Closed contours must completely lie inside the workpiece blank definition. Otherwise, the system will follow closed contours also along the rotary axis when machining, causing collisions.

- Select or program only those contour elements that are actually required (for example, within the definition of a finished part).

To select a closed contour:

- Select **Contour**
- Select all required contour elements
- Select the starting point of the first element
- > The control closes the contour.

14.4.2 Selecting positions

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 89
- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.
  Further information: "Selecting and saving a contour", Page 351

Individual selection

To select individual positions (e.g., holes):

- Select **Positions**
- Position the cursor on the desired element
- > The control shows the circumference and center point of the element in orange.
- Select the desired element
- > The control highlights the selected element in blue and displays it in the List View area.

Multiple selection within an area

To select multiple positions within an area:

- Select **Positions**
- Drag a box around the area while holding down the left mouse button
- > The control opens the **Find circle centers by diameter range** window. The window shows the identified diameter and depth values.
- Change the filter settings as needed
- Select **OK**
- > The control loads all positions within the selected diameter and depth ranges into the List View area.
- > The control shows the traverse distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:

- Select **Positions**

- Select **Find circles according to diameter range. Load center coordinates to the position list**
- > The control opens the **Find circle centers by diameter range** window. The window shows the identified diameter and depth values.

Notes

- Set the correct unit of measure so that **CAD Viewer** shows the correct values.
- Ensure that the unit of measure used in the NC program matches that used in **CAD Viewer**. Elements that have been copied from **CAD Viewer** to the clipboard do not contain any information about the unit of measure.
- The control retains the content of the clipboard only as long as **CAD Viewer** is open.
- **CAD Viewer** also considers circles that consist of two semicircles to be one machining position.
- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.
- CAD Import outputs the radii of the circular arcs as comments. At the end of the generated NC blocks, CAD Import displays the smallest radius to help you select the most suitable tool.

Notes on Contour Transfer

- If you double-click a layer in the List View area, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved it on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.

  As an alternative, select the Remove mode.

14.5 Generating STL files with 3D mesh (#152 / #1-04-1)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective fixture and tool holder files, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture management
- Export the simulated workpiece as an STL file
- Using an STL file as workpiece blank
  Further information: Programming and Testing User's Manual

Requirement

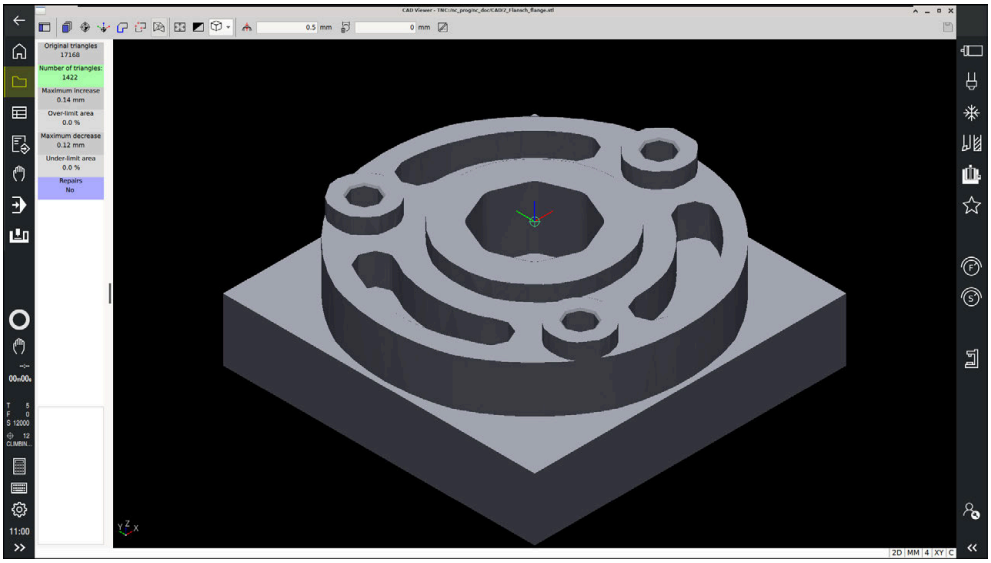
- Software option CAD Model Optimizer (#152 / #1-04-1)

Description of function

When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.



3D model in **3D mesh** mode

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the settings selected in **3D mesh** mode.

The List View area shows the following information:

Option	Meaning
Original triangles	Number of triangles in the original model

Option	Meaning
Number of triangles:	Number of triangles with active settings in the simplified model If this option is highlighted in green, the number of triangles is in the optimum range. You can further reduce the number of triangles using the available functions. **Further information:** "Functions for the simplified model", Page 358
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control indicates the type of repair (e.g., **Hole Int Shells**). This indication consists of the following items: ■ **Hole** **CAD Viewer** closed holes in the 3D model. ■ **Int** **CAD Viewer** removed self-intersections. ■ **Shells** **CAD Viewer** joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:

- Max. 20 000 triangles
- Triangular mesh forms a closed shell

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD Viewer provides the following functions:

Icon	Meaning
	**Allowed simplification** Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	**Remove holes <= diameter** Use this function to remove holes and pockets up to the specified diameter from the original model.
	**Only optimized mesh shown** The control shows the simplified model only.
	**Original is displayed** The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	**Save** Use this function to save the simplified 3D model with the selected settings as an STL file.

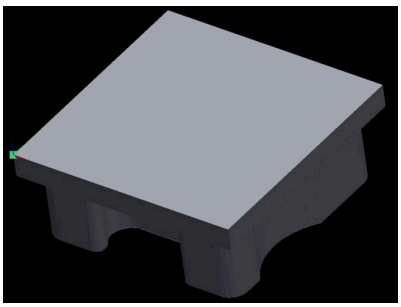
14.5.1 Positioning the 3D model for rear-face machining

To position an STL file for rear-face machining:

- Export the simulated workpiece as an STL file
  Further information: Programming and Testing User's Manual

- Select the **Files** operating mode
- Select the exported STL file
- > The control opens the STL file in **CAD Viewer**.

- Select **Preset**
- > The control displays information on the preset position in the List View area.
- Enter the value of the new preset in the **Preset** area (e.g., **Z-40**)
- Confirm your input
- Orient the coordinate system by specifying values under **PLANE SPATIAL SP*** (e.g., **A+180** and **C+90**)
- Confirm your input



- Select **3D mesh**
- > The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- Further simplify the 3D model using the **3D mesh** mode functions, if required.
  Further information: "Functions for the simplified model", Page 358

- Select **Save**
- > The control opens the **Define file name for 3D mesh** window.
- Enter the desired name
- Select **Save**
- > The control saves the STL file positioned for rear-face machining.

The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.
Further information: Programming and Testing User's Manual

15

User aids

15.1 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.

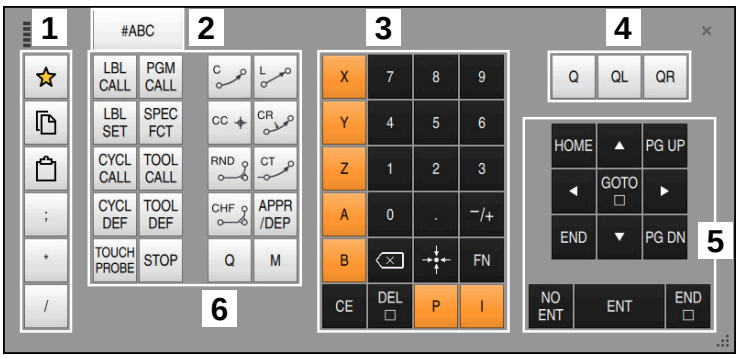
You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

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

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**

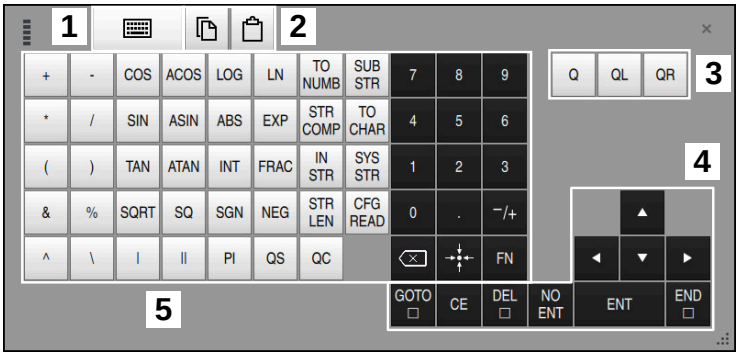
Text input areas

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

15.1.1 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.

15.2 Message menu on the information bar

Application

In the message menu of the information bar, the control shows pending errors and notes. When opened, the control displays detailed information about the messages.

Description of function

The control uses the following symbols to differentiate between the types of messages:

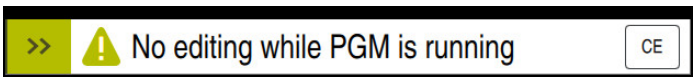
Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot clear this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
	Error Reset type	The control must be restarted. This message cannot be cleared.
	Error Emergency-stop type	The control performs an emergency stop. An error message can only be cleared after the cause has been eliminated.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content



Collapsed message menu on the information bar

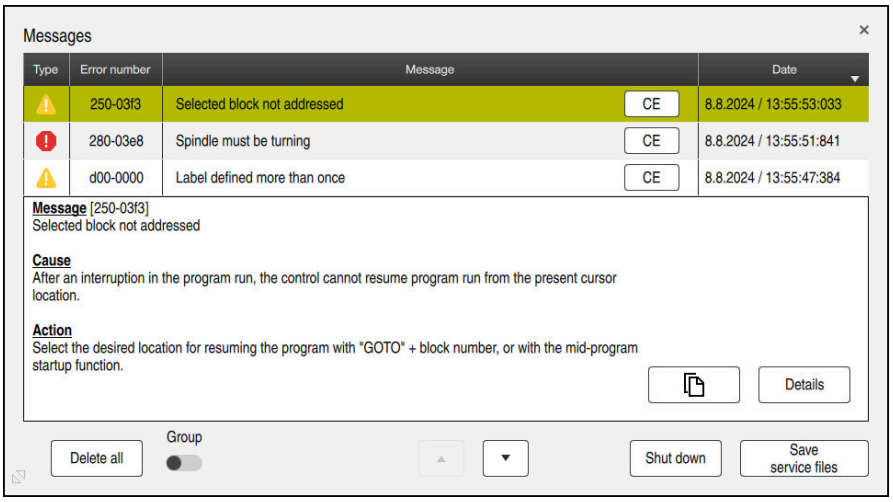
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (root cause, correction, information on the NC program)

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** toggle switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

- If an error occurs, you can create a service file manually.
  Further information: "Creating a service file manually", Page 367
- If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.
  Further information: "Creating a service file automatically", Page 368

Service files help service technicians in troubleshooting the problem. The control saves data that provide information about the current machine and operation status, such as active NC programs up to 10 MB, tool data, and keystroke logs.

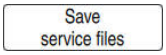
The file name of each service file consists of a user-defined name and a timestamp.

If you create multiple service files with the same name, the control saves a maximum of five files and then deletes the file with the oldest timestamp, if necessary. Make a backup of the service files you created (e.g., by moving them to a different folder).

15.2.1 Creating a service file manually

To create a service file manually:

- ▶ Expand the message menu



- ▶ Select **Save service files**
- > The control opens the **Save service file** window.
- ▶ Enter the file name

- ▶ Press **OK**
- > The control saves the service file in the **TNC:\service** directory.

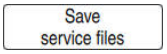
Using a toggle switch, you can define whether the control will save data from process monitoring (#168 / #5-01-1) for the current NC program in the service file.

15.2.2 Creating a service file automatically

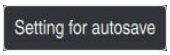
You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

To specify a new error number:

- Expand the message menu



- Select **Save service files**
- > The control opens the **Save service file** window.



- Select **Setting for autosave**
- > The control opens a table of error numbers.
- Enter the desired error number
- Enable the **Active** check box
- > If the error occurs, the control automatically creates a service file.
- Enter a comment, if applicable (e.g., to describe the problem)

16

The MDI Application

Application

The **MDI** application allows you to execute individual NC blocks outside of the context of an NC program (e.g., **PLANE RESET**). When you press the **NC Start** key, the control will run the NC blocks separately.

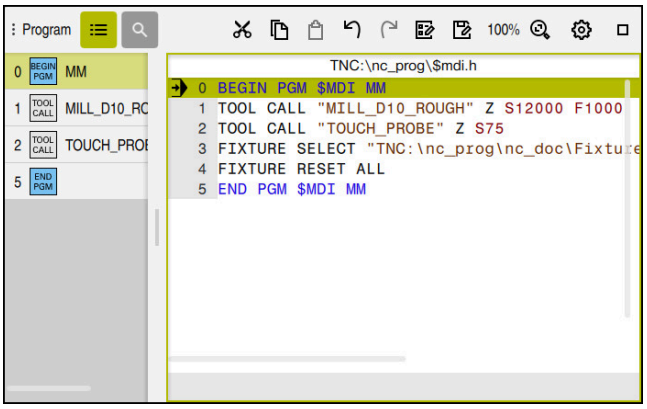
You can also create an NC program step by step. The control remembers modally effective program information.

Related topics

- Creating NC programs
  Further information: Programming and Testing User's Manual
- Running NC programs
  Further information: "Program run", Page 411

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **$mdi_inch.h**.



The **Program** workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **GPS** (#44 / #1-06-1)
  Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287
- **Help**
- **Positions**
  Further information: "The Positions workspace", Page 119
- **Program**
  Further information: Programming and Testing User's Manual
- **Simulation**
  Further information: Programming and Testing User's Manual
- **Status**
  Further information: "The Status workspace", Page 127
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 362

Icons and buttons

In the **MDI** application, the function bar provides the following buttons:

Icon or button	Meaning
	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Klartext editor	If this toggle switch is 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:** Programming and Testing User's Manual
Insert NC function	The control opens the **Insert NC function** window. **Further information:** Programming and Testing User's Manual
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:** Programming and Testing User's Manual
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** Programming and Testing User's Manual
/ 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:** Programming and Testing User's Manual
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual
; 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:** Programming and Testing User's Manual
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 417
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC (#145 / #2-30-1)). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 286
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. **Further information:** User's Manual for Machining Cycles
Edit	The control opens the context menu. **Further information:** Programming and Testing User's Manual
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 197
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 418

Icon or button	Meaning
Reset program	If you select **Internal stop**, the control activates this button. The control resets any modally active program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always execute NC blocks in **Single Block** mode. After the control has executed an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 418

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Notes

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the **Q parameter list** window
  - Switching the operating modes
- Restore the contextual reference by repeating the required NC blocks

- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
- The following functions are not available in the **MDI** application:
  - Calling of an NC program with **PGM CALL**
  - Test run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** functions while program run is interrupted
  - **Block scan** function
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

17

Touch probe functions in the Manual operating mode

17.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Related topics

- Automatic touch probe cycles for the workpiece
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Preset table
  Further information: "Preset table *.pr", Page 503
- Datum table
  Further information: Programming and Testing User's Manual
- Reference systems
  Further information: "Reference systems", Page 214
- Preassigned variables
  Further information: Programming and Testing User's Manual

Requirements

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 391

Description of function

The control provides the following functions for setting up the machine in the **Setup** application of the **Manual** operating mode:

- Define the workpiece preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- **Measure the tool**
- **Set up fixtures** (#140 / #5-03-2)

  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256
- **Set up the workpiece** (#159 / #1-07-1)

  Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398

Within the functions, the control provides the following probing methods:

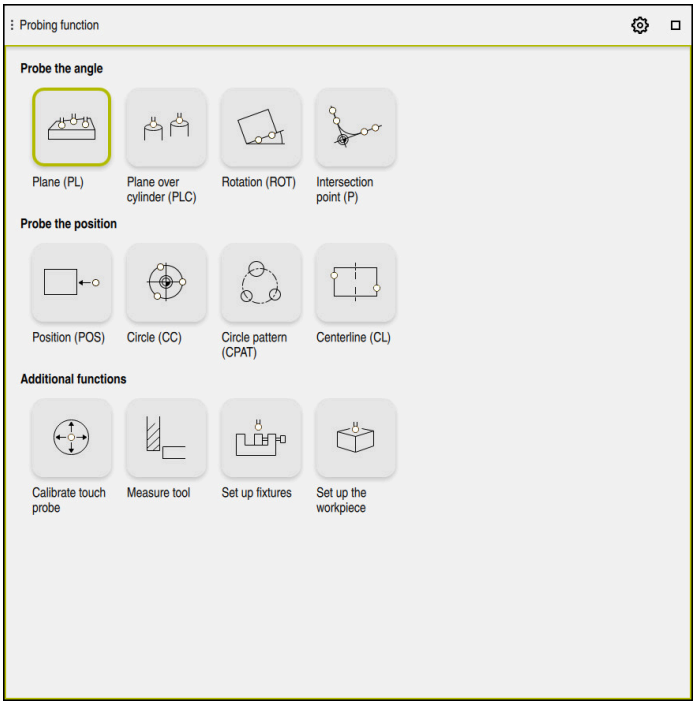
- Manual probing method

  You position and start individual probing processes manually within a touch probe function.

  Further information: "Setting a preset in a linear axis", Page 384
- Automatic probing method

  You manually position the touch probe to the first probing point before the start of the probing routine and fill out a form with the individual parameters for the respective touch probe function. When you start the touch probe function, the control automatically positions and automatically performs probing.

  Further information: "Determining the circle center point of a stud using the automatic probing method ", Page 385



The **Probing function** workspace

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the **Plane (PL)** function to determine the solid angle of a plane. You then save the values in the preset table or align the plane.
Plane over cylinder (PLC) 	Use the **Plane over cylinder (PLC)** function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table or align the plane.
Rotation (ROT) 	Use the **Rotation (ROT)** function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. **Further information:** "Determining and compensating the basic rotation of a workpiece", Page 387
Intersection point (P) 	Use the **Intersection point (P)** function to probe four probing objects. The probing objects can be either positions or circles. The control determines the intersection of the axes and the skew of the workpiece from the objects that have been probed. You can set the intersection point as a preset. You can transfer the determined skew to the preset table as a basic transformation or as an offset.

The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.

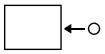
Further information: "Preset table *.pr", Page 503

You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

Further information: "Comparison of offset and 3D basic rotation", Page 409

Probe the position

The **Probe the position** group contains the following touch probe functions:

Button	Function
Position (POS) 	You can use the **Position (POS)** function to probe a position in the X axis, Y axis or Z axis. **Further information:** "Setting a preset in a linear axis", Page 384
Circle (CC) 	The **Circle (CC)** function is used to determine the coordinates of a circle center point (e.g., for a hole or for a stud). **Further information:** "Determining the circle center point of a stud using the automatic probing method ", Page 385
Circle pattern (CPAT) 	The **Circle pattern (CPAT)** function is used to determine the center point coordinates of a circle pattern.
Centerline (CL) 	The **Centerline (CL)** function is used to determine the center point of a ridge or slot.

Additional functions

The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe 	The **Calibrate touch probe** function is used to determine the length and radius of a workpiece touch probe. **Further information:** "Calibrating the workpiece touch probe", Page 391
Measure tool 	The **Measure tool** function allows you to measure tools by scratching. In this function, the control supports milling tools, drilling tools and turning tools. **Further information:** "Werkzeug vermessen mit Ankratzen", Page
Set up fixtures 	The **Set up fixtures** function is used to determine the position of a clamping device in the working space using a workpiece touch probe (#140 / #5-03-2). **Further information:** "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256
Set up the workpiece 	The **Set up the workpiece** function is used to determine the position of a workpiece in the working space using a workpiece touch probe (#159 / #1-07-1). **Further information:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 398

Icons and buttons

General icons and buttons in the touch probe functions

The following icons and buttons are available, depending on the selected touch probe function:

Icon or button	Meaning
	**Exit probing**
	Select the workpiece preset and the pallet preset and edit the values if required The control shows the number of the active preset to the right of the icon. **Further information:** "The Change the preset window", Page 382 **Further information:** "Preset table *.pr", Page 503
	Display help graphics for the selected touch probe function
	Select the probing direction
	**actual position capture**
	Manually approach and probe points on a straight surface
	**Measuring method M** Manually approach and probe points on a stud or in a hole
	**Measuring method A** Automatically approach and probe points on a stud or in a hole After the last touching process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 197
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 418

Icons and buttons for calibration

The control offers the following functions for calibrating a 3D touch probe:

Icon or button	Meaning
	Calibrating the length of a 3D touch probe
	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 391

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Icon	Meaning
	Measure the radius and the center offset using a calibration ring
	Measure the radius and the center offset using a stud or a calibration pin
	Measure the radius and the center offset using a calibration sphere Optional 3D calibration of workpiece touch probe (#92 / #2-02-1) **Further information:** Programming and Testing User's Manual **Further information:** "3D calibration (#92 / #2-02-1)", Page 392

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **Manual operation** and **Setup** applications, the control opens the **Working plane is inconsistent!** window. The status of the tilting situation is shown in the **3-D rotation** window.

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 239

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Meaning
3-D ROT Apply status	■ If the positions of the rotary axes match the tilting situation in the program run (**Program Run** operating mode and **MDI** application), the control will apply this tilting situation. ■ Otherwise the control will apply the current rotary-axis angles as the tilting angles. The **3-D rotation** window shows the applied spatial or axis angles. **Further information:** "The 3-D rotation window (#8 / #1-01-1)", Page 239
3-D ROT Ignore status	The control ignores the tilting situation and moves the axes as if the rotary axes were at their zero position.
Align the rotary axes	The control positions the rotary axes in a manner suitable to the current tilting situation. If, for example, no tilting function is active and the rotary axes have been moved, the control then returns the rotary axes to their zero position.

Buttons for touch-probe functions

The control offers these buttons in the touch-probe functions during or after probing:

Button	Meaning
Remove last measurement	The **Remove last measurement** function causes the control to undo the last probe point. The control offers this button only when using measuring method **M**.
Assume measurement result	The **Assume measurement result** function applies the measuring result for the current measurement. The control offers this button only when using measuring method **M**.
Compensate the active preset	The **Compensate the active preset** function transfers the measuring result into the active line of the preset table. **Further information:** "Preset table *.pr", Page 503
Correct the datum	The **Correct the datum** function transfers the measuring result into a desired row of the datum table. **Further information:** Programming and Testing User's Manual
Align rotary table	The **Align rotary table** function aligns the rotary axes mechanically according to the measuring result.
Correct the pallet reference point	The **Correct the pallet reference point** function transfers the measuring result into the active line of the pallet preset table. **Further information:** Programming and Testing User's Manual

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets
- Change pallet presets only after discussion with the machine manufacturer
- Check the pallet preset in the **Setup** application before you start machining

The Change the preset window

In the **Change the preset** window you can select a preset or edit the values of a preset.

Further information: "Preset management", Page 230

The **Change the preset** window provides the following buttons:

Icon or button	Meaning
	The control shows the preset table. **Further information:** Programming and Testing User's Manual
	The control shows the pallet preset table. **Further information:** Programming and Testing User's Manual
Reset basic rotation	The control resets the values from the columns **SPA**, **SPB** and **SPC**.
Reset offsets	The control resets the values from the columns **A_OFFS**, **B_OFFS** and **C_OFFS**.
Apply changes and delete existing probe objects	The control activates the selected preset and rejects the touch points used so far. Then the control closes the window.
Apply	The control saves the changes and the selected preset. Then the control closes the window.
Reset	The control cancels the changes and restores the initial condition.
Cancel	The control closes the window without saving.

If you change a value, the control marks this value with a blue dot.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets
- Change pallet presets only after discussion with the machine manufacturer
- Check the pallet preset in the **Setup** application before you start machining

Log file of touch probe cycles

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

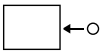
If you run several touch probes cycles in a row, the control stores the measured values below each other.

17.1.1 Setting a preset in a linear axis

To probe the preset in any axis:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool

- Select the **Setup** application

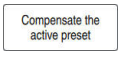


- Select the **Position (POS)** touch probe function
- > The control opens the **Position (POS)** touch probe function.

- If necessary, check the values of the active preset or choose a different preset
- Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)

- Select the probing direction (e.g., **Z-**)

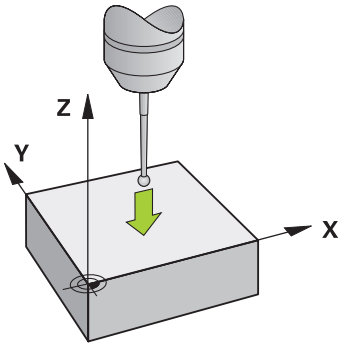
- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)



- Select **Compensate the active preset**
- > The control enters the defined nominal value in the preset table.

> **Using the Position (POS) probing function, you can probe in up to three axes.**

- Select **Exit probing**
- > The control closes the **Position (POS)** probing function.



17.1.2 Determining the circle center point of a stud using the automatic probing method

To probe a circle center point:

- Select the **Manual** operating mode

- Call the workpiece touch probe as a tool
- Select the **Setup** application

- Select **Circle (CC)**

> The control opens the **Circle (CC)** probing function.

- If necessary, check the values of the active preset or choose a different preset

- Select **Measuring method A**

- Select **Type of contour** (e.g., stud)
- Enter **Diameter** (e.g., 60 mm)
- Enter **Safety clearance (min. value = SET_UP)** if required

> The control suggests the total of the value in the **SET_UP** column of the touch probe table and the ball tip radius as a safety distance.

- Enter **Starting angle** (e.g., –180°)
- Enter **Angular length** (e.g., 360°)
- Position the 3D touch probe at the desired probing position next to the workpiece and below the workpiece surface
- If necessary, turn the feed rate potentiometer to zero

- Press the **NC Start** key

- If necessary, slowly turn up the feed-rate potentiometer

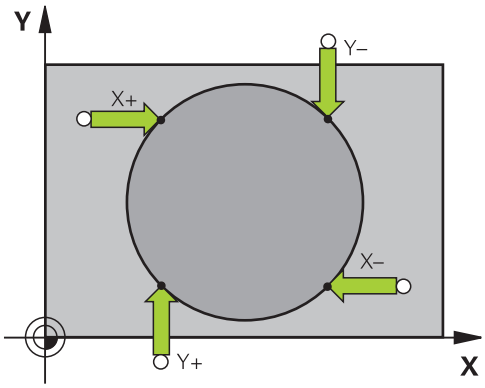
> The control executes the touch probe function based on the data entered.

> The control shows the measurement results.

- In the **Nominal value** area, enter the new preset of the scanned axes (e.g., **0**)

Compensate the active preset

- ▶ Select **Compensate the active preset**
- > The control sets the preset to the entered nominal value.
- ▶ Select **Exit probing**
- > The control closes the **Circle (CC)** probing function.



17.1.3 Determining and compensating the basic rotation of a workpiece

To probe the basic rotation of a workpiece:

- Select the **Manual** operating mode

- Call the 3D touch probe as a tool
- Select the **Setup** application

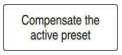
- Select **Rotation (ROT)**
> The control opens the **Rotation (ROT)** probing function.

- If necessary, check the values of the active preset or choose a different preset
- Position the 3D touch probe at the desired probing position in the workspace

- Select the probing direction (e.g., **Y+**)

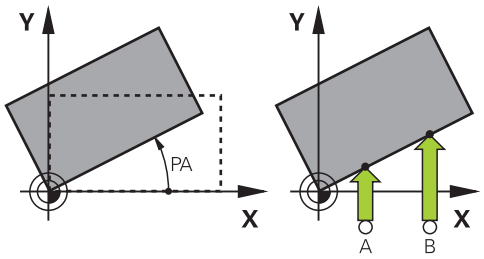
- Press the **NC start** key
> The control executes the first probing process and limits the subsequently selectable probing directions.
- Position the 3D touch probe at the second probing position in the workspace

- Press the **NC start** key
> The control executes the probing process and then shows the measurement results.
- Activate the **Basic rotation** toggle switch as needed



- Select **Compensate the active preset**
> The control transfers the determined basic rotation with respect to the tool axis to the **SPC** column of the active line of the preset table.

- Select **Exit probing**
> The control closes the **Rotation (ROT)** probing function.



Instead of **Basic rotation** you can also select the **Table rotation** toggle switch. The control then transfers the values into columns **A_OFFS**, **B_OFFS** or **C_OFFS**.

17.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can use all manual touch probe functions with manual probing methods with mechanical buttons or with scratching.

For this, the control provides the **Accept position** button.

To determine a basic rotation with a mechanical probe:

- Select the **Manual** operating mode

- Insert the tool, such as an analog 3D probe or feeler lever gage
- Select the **Setup** application

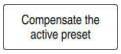
- Select the **Rotation (ROT)** probing function

- If necessary, check the values of the active preset or choose a different preset

- Select the probing direction (e.g., **Y+**)
- Move the mechanical probe to the first position to be captured by the control.

- Select **Accept position**
> The control saves the current position.
- Move the mechanical probe to the next position to be captured by the control.

- Select **Accept position**
> The control saves the current position.
- Activate the **Basic rotation** toggle switch as needed



- Select **Compensate the active preset**
> The control transfers the determined basic rotation with respect to the tool axis to the **SPC** column of the active line of the preset table.

- Select **Exit probing**
> The control closes the **Rotation (ROT)** probing function.

Instead of **Basic rotation** you can also select the **Table rotation** toggle switch. The control then transfers the values into columns **A_OFFS**, **B_OFFS** or **C_OFFS**.

Notes

- If you use a non-contacting tool touch probe (such as a laser touch probe), then you are using touch-probe functions from a third-party supplier. Refer to your machine manual.
- The accessibility of the pallet preset table in the touch-probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- The use of touch-probe functions deactivates the Global Program Settings (GPS) (#44 / #1-06-1) temporarily.

  Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287
- You can use the manual touch-probe functions only with restrictions in turning mode (#50 / #4-03-1).
- You must calibrate the touch probe separately in turning mode. The factory default setting of the worktable may vary between milling mode and turning mode, which is why you must calibrate the touch probe without any center offset in turning mode. You can create a tool index for storing the additionally calibrated tool data in the same tool.

  Further information: "Indexed tool", Page 188
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no tool touch probe is inserted, the actual position can be captured with **NC START**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
  - Initial configuration
  - Broken stylus
  - Stylus replacement
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis
- If the touch point is not reached during the touching process, the control will display a warning. The probing process can be continued with **NC Start**.
- The machine manufacturer use the optional machine parameter **trackAsync** (no. 122503) to define whether the control orients the spindle during pre-positioning for probing. This can save time during automatic probing procedures. Additionally, the control takes the calibrated center offset of L-shaped styli into account for the spindle tracking speed. This means that the speed at the ball tip is at most the rapid traverse of the probe **FMAX**, which increases safety during probing.

Definition

Spindle tracking
If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

17.2 Calibrating the workpiece touch probe

Application

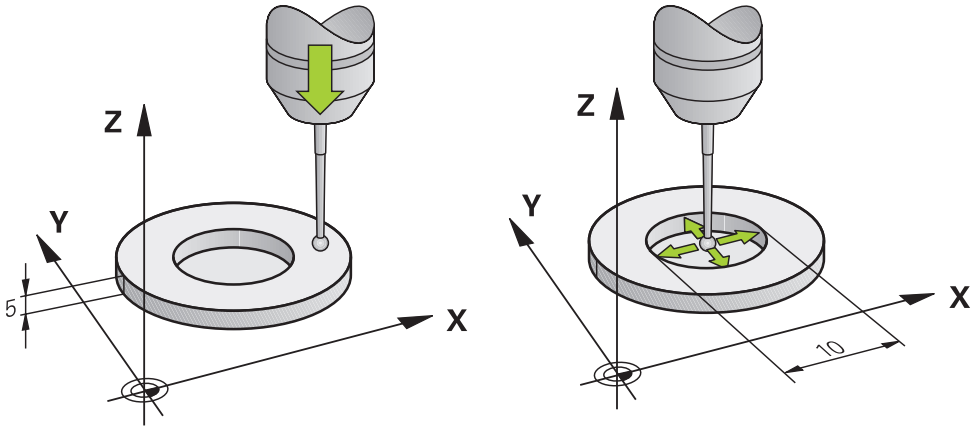
The touch probe must be calibrated in order to enable determining precisely the actual switching point of a 3D touch probe. Otherwise, the control cannot ascertain exact measuring results.

During 3D calibration, the angle-dependent deflection behavior of a workpiece touch probe is determined in any probing direction (#92 / #2-02-1). Even if there is no exact radial or axial deflection of the workpiece touch probe, you will obtain precise measuring results with the 3D calibration.

Related topics

- Calibrate the workpiece touch probe automatically
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Touch probe table
  Further information: "Touch probe table tchprobe.tp", Page 489
- Tool angle-dependent 3D radius compensation (#92 / #2-02-1)
  Further information: Programming and Testing User's Manual

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 179

You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

At first, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis For this, manually position the extension in the direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp", Page 489

When calibrating the length, the control aligns the touch probe with the calibration angle defined in the **CAL_ANG** column.

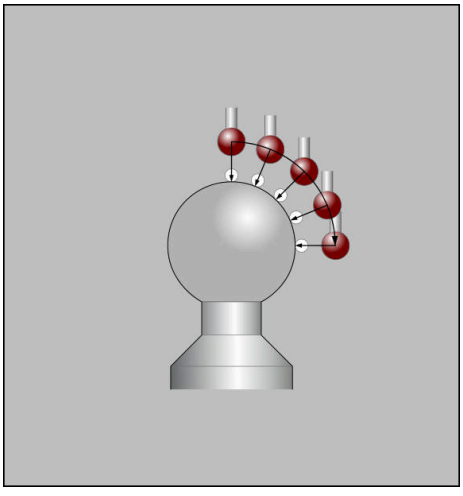
While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

3D calibration (#92 / #2-02-1)

In addition to calibrating with a calibration sphere, the control also enables the touch probe to be calibrated dependent on the angle. For this purpose the control probes the calibration sphere in a quarter circle in the perpendicular. The 3D calibration data specifies the deflection behavior of the touch probe in any probing direction.

The control saves the deviations in a compensation value table ***.3DTC** in the folder **TNC:\system\3D-ToolComp**.

The control creates a specific table for each calibrated touch probe. In the tool table the **DR2TABLE** column is automatically referenced to this.



3D calibration

Reversal measurement

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

HEIDENHAIN touch probes are predefined as to whether or how a touch probe can be oriented. Other touch probes are configured by the machine manufacturer.

When calibrating the radius, up to three circular measurements can be taken depending on the possible orientation of the workpiece touch probe. The first two circular measurements determine the center offset of the workpiece touch probe. The third circular measurement determines the effective stylus tip radius. If orientation of the spindle is not possible or only a certain orientation is possible due to the workpiece touch probe, circular measurements are omitted.

17.2.1 Calibrating the length of the workpiece touch probe

To calibrate a workpiece touch probe using an overmilled surface in length:

- Measure the end milling cutter on the tool presetting device
- Store the measured end milling cutter in the tool magazine of the machine
- Enter the tool data of the end milling cutter in tool management
- Clamp the workpiece blank

- Select the **Manual** operating mode
- Replace the end milling cutter in the machine
- Switch on spindle (e.g., with **M3**)
- Use the handwheel to scratch the workpiece blank

  Further information: "Setting a preset with milling cutters", Page 231
- Set preset in the tool axis (e.g., with **Z**)
- Position the end milling cutter next to the workpiece blank
- Set a small value in the tool axis (e.g., **-0.5 mm**)
- Overmill the workpiece blank using the handwheel
- Set the preset again in the tool axis (e.g., with **Z=0**)
- Switch off spindle (e.g., with **M5**)
- Replace the tool touch probe
- Select the **Setup** application

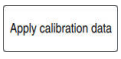
- Select **Calibrate touch probe**

- If necessary, check the values of the active preset or choose a different preset

- Select the **Length calibration** measurement method
- The control displays the current calibration data.
- Enter the reference surface position (e.g., with **0**)
- Position the workpiece touch probe close to the surface of the overmilled area

> Check that the area to be probed is flat and free of chips before you start the touch probe function.

- Press the **NC Start** key
- The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- Check results



- Select **Apply calibration data**
- The control transfers the calibrated length of the 3D touch probe to the tool table.

- Select **Exit probing**
- The control closes the **Calibrate touch probe** function.

17.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a setting ring in the radius:

- Clamp the setting ring on the machine table (e.g., with clamps)

- Select the **Manual** operating mode
- Position the 3D touch probe in the hole of the setting ring

> Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe with the largest point of the stylus tip.

- Select the **Setup** application

- Select **Calibrate touch probe**

- If necessary, check the values of the active preset or choose a different preset

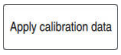
- Select the **Radius** measurement method

- Select the **Setting ring** calibration standard
- Enter the diameter of the ring gauge
- Enter the start angle
- Enter the number of touch points

- Press the **NC Start** key

> The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

- Check results



- Select **Apply calibration data**

> The control stores the calibrated radius of the 3D touch probe in the tool table.

- Select **Exit probing**

> The control closes the **Calibrate touch probe** function.

17.2.3 3D calibration of workpiece touch probe (#92 / #2-02-1)

To calibrate a workpiece touch probe using a calibration sphere in the radius:

- Clamp the setting ring on the machine table (e.g., with clamps)

- Select the **Manual** operating mode
- Position the workpiece touch probe centrally above the sphere
- Select the **Setup** application

- Select **Calibrate touch probe**

- If necessary, check the values of the active preset or choose a different preset

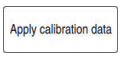
- Select **Radius** measurement method

- Select the **Calibration sphere** calibration standard
- Enter the diameter of the sphere
- Enter the start angle
- Enter the number of touch points

- Press the **NC Start** key

> The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

- Check results



- Select **Apply calibration data**

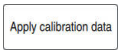
> The control stores the calibrated radius of the 3D touch probe in the tool table.

> The control shows the **3D calibration** measurement method.

- Select the **3D calibration** measurement method
- Enter the number of touch points

- Press the **NC Start** key

> The 3D touch probe probes all required touch points in an automatic probing routine.



- Select **Apply calibration data**

> The control saves the deviations in a compensation value table under **TNC:\system\3D-ToolComp**.

- Select **Exit probing**

> The control closes the **Calibrate touch probe** function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

  Further information: "Touch probe table tchprobe.tp", Page 489

17.3 Setting up the workpiece with graphical support (#159 / #1-07-1)

Application

Use the **Set up the workpiece** function to determine the position and misalignment of a workpiece with only one touch-probe function and save it as a workpiece preset. During setup, you can probe curved surfaces.

The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model.

Related topics

- Touch-probe functions in the **Setup** application

  Further information: "Touch probe functions in the Manual operating mode", Page 373
- Generating an STL file of a workpiece

  Further information: Programming and Testing User's Manual
- **Simulation** workspace

  Further information: Programming and Testing User's Manual
- Setting fixtures with graphical support (#140 / #5-03-2)

  Further information: "Integrating fixtures into collision monitoring (#140 / #5-03-2)", Page 256

Requirements

- Model Aided Setup (#159 / #1-07-1) software option
- Touch probe properly defined in the tool management:
  - Sphere radius in parameter **R2**
  - If probing on inclined surfaces, spindle tracking must be active in the **TRACK** parameter

  Further information: "Parameters of the touch probe table tchprobe.tp", Page 490
- Workpiece touch probe calibrated

  If probing on inclined surfaces, HEIDENHAIN recommends performing a 3D calibration of the touch probe (#92 / #2-02-1).

  Further information: "Calibrating the workpiece touch probe", Page 391
- NC program with **BLK FORM** workpiece blank definition or STL file of the workpiece blank

Description of function

The control compares the positions of the probe points on the actual workpiece with a 3D model of the workpiece.

The more the 3D model corresponds to the actual workpiece, the higher the possible workpiece setup accuracy.

You have the following options for defining the 3D model:

- **BLK FORM** in the NC program

  The control creates the 3D model using the workpiece blank definition.
- STL file of the workpiece with up to 300 000 triangles

The scope of the **Set up the workpiece** function depends on the Adv. Function Set 1 (#8 / #1-01-1) and Adv. Function Set 2 (#9 / #4-01-1) software options as follows:

- Both software options enabled:

  You can tilt before setting up and incline the tool while setting up in order to probe even complex workpieces (e.g., shaped parts).
- Only Adv. Function Set 1 (#8 / #1-01-1) is enabled:

  You can tilt before setting up. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.

  > If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.
- None of the two software options is enabled:

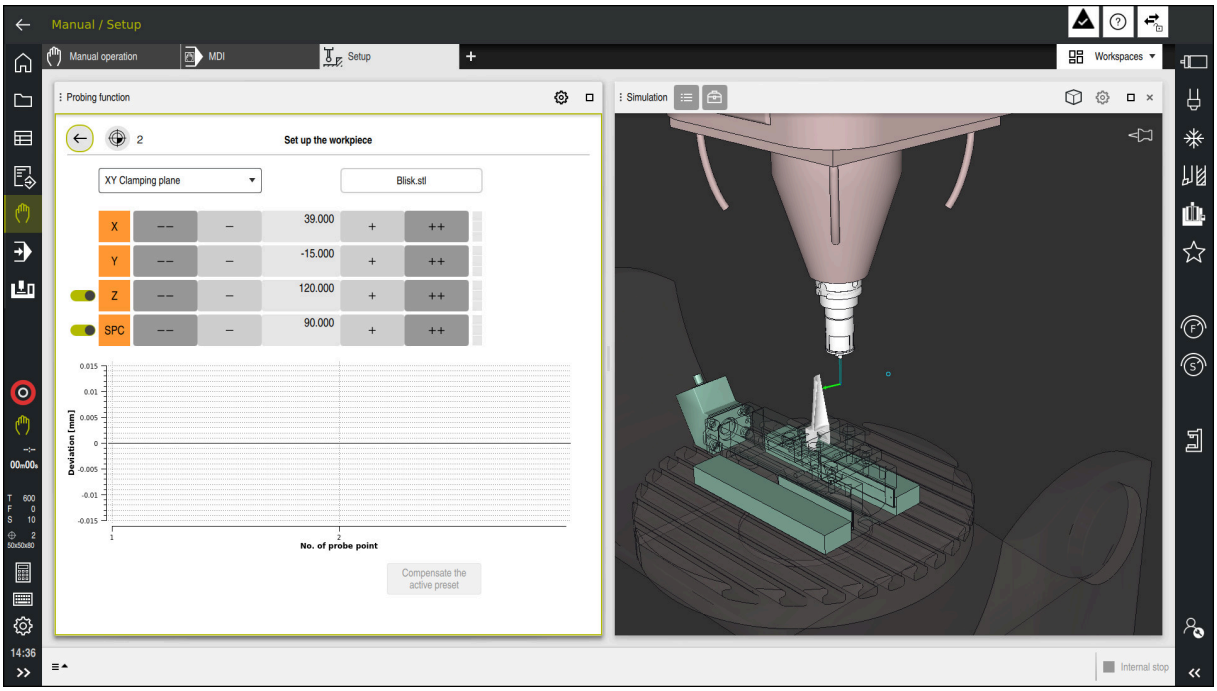
  You cannot tilt before setting up. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 235

Further information: Programming and Testing User's Manual

Extension of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphical support for setting up the workpiece.



The **Set up the workpiece** function with the **Simulation** workspace open

When the **Set up the workpiece** function is active, the **Simulation** workspace displays the content below:

- Current position of workpiece as viewed by the control
- Probed points on the workpiece
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the workpiece or the workpiece touch probe is positioned within the workpiece, as seen by the control.

    In this case you can correct the position of the 3D model in the simulation, if required.
  - Red arrow

    Probing in the direction of the arrow is not possible.

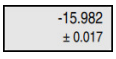
    > Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    Probing in the direction of the arrow is possible to a limited extent. The probing is performed in a deselected direction or could cause collisions.
  - Green arrow

    Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up the workpiece** function contains the following icons and buttons:

Icon or button	Meaning
	Open the **Change the preset** window You can select the workpiece preset and the pallet preset and edit values if required. The control shows the number of the active preset to the right of the icon. After the first point has been probed, the control dims the icon.
XY Clamping plane	Use this selection menu to define the probing mode. Depending on the probing mode, the control displays the respective axis directions and spatial angles. **Further information:** "Probing mode", Page 402
	File name of the NC program or 3D model
	Shifts the position of the virtual workpiece by 10 mm, 0.3937 inches, or 10° in the negative axis direction Shifts the workpiece in mm or inches in a linear axis, and in degrees in a rotary axis.
	Shifts the position of the virtual workpiece by 1 mm, 0.0394 inches, or 1° in the negative axis direction
	■ Enter the position of the virtual workpiece directly ■ Value and estimated accuracy of the value after the probing
	Shifts the position of the virtual workpiece by 1 mm, 0.0394 inches, or 1° in the positive axis direction
	Shifts the position of the virtual workpiece by 10 mm, 0.3937 inches, or 10° in the positive axis direction
    	Status of the direction: ■ Dimmed The axis direction is deselected in this setup process and is not considered. ■ Empty No touch points have been determined yet. ■ Red The control cannot locate the workpiece position in this axis direction. ■ Yellow The position of the workpiece in this axis already contains information. The information is not meaningful yet. ■ Green The control can locate the workpiece position in this axis direction.
Compensate the active preset	The control saves the determined values in the active row of the preset table.

Probing mode

The following modes for probing the workpiece are available to you:

- **XY Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPC**
- **XZ Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPB**
- **YZ Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPA**
- **6D**
  X, **Y** and **Z** axis directions as well as spatial angles **SPA**, **SPB** and **SPC**

Depending on the probing mode, the control displays the respective axis directions and spatial angles. In the **XY**, **XZ** and **YZ** clamping planes a toggle switch allows you to deselect the respective tool axis and spatial angle, if required. The control will not take deselected axis directions into account in the setup process and positions the workpiece by considering the remaining axis directions only.

HEIDENHAIN recommends executing the setup process as follows:

1 Pre-position a 3D model in the machine's working space

  By default, the control positions the 3D model to the active workpiece reference point. At this point in time, the control does not know the precise position of the workpiece, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe produces values close to the position of the real workpiece.

2 Set the first touch points in the **X**, **Y** and **Z** axis directions

  If the control can determine the position in one axis direction, it will change the status of that axis to green.

3 Determine the spatial angle by setting further touch points

  To achieve maximum accuracy when probing the spatial angles, the touch points should be as far apart from one another as possible.

4 Increase the accuracies by additional check points

  Additional check points at the end of the measuring process improve the matching accuracy and minimize the misalignment between the 3D model and the real workpiece. Perform as many probing processes as necessary until the control displays the desired accuracy beneath the current value.

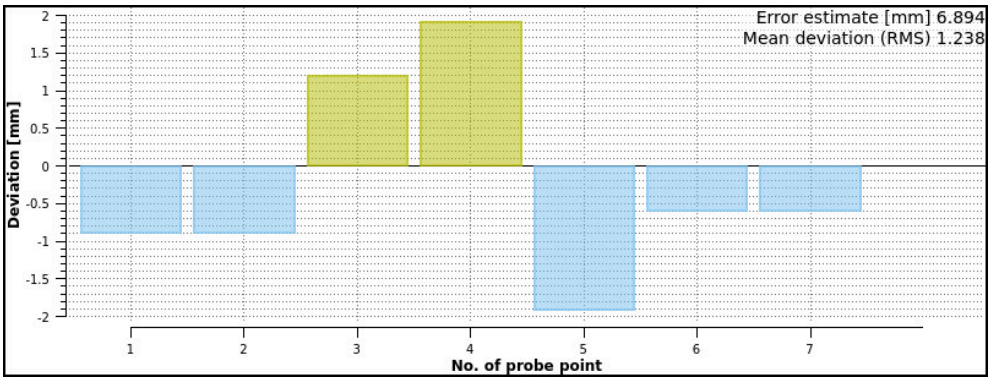
The error estimate diagram shows for each probe point the distance of the 3D model from the real workpiece.

Further information: "Error estimate diagram", Page 403

Error estimate diagram

Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows for each probe point the distance of the 3D model from the real workpiece.



Error estimate diagram in the **Set up the workpiece** function with transparent columns

The error estimate diagram of the **Set up the workpiece** function shows the following information:

- **Error estimate [mm]**

  After each probe point, the value shows the greatest estimated distance between the 3D model and the workpiece.
- **Mean deviation (RMS)**

  After each probe point, the value shows the average of all measured distances between the 3D model and the workpiece.
- **Deviation [mm]**

  Using this axis you can determine how large the estimated distance between the 3D model and the probe point on the workpiece is.
- **No. of probe point**

  This axis shows the numbers of the probe points so far.
- Columns

  As long as the status of all axes is not green, the control shows transparent columns.

  The control recalculates the 3D model after each probe point. This also changes the previous values.

  When the columns of the error estimate diagram are no longer transparent, and **Error estimate [mm]** shows the desired accuracy, the setup process is complete.

The following factors influence the accuracy that can be achieved when measuring workpieces:

- Accuracy of workpiece touch probe
- Accuracy of the machine kinematic configuration
- Deviations of the 3D model from the real workpiece
- Condition of the actual workpiece (e.g., unmachined areas)

17.3.1 Setting up a workpiece

Use the **Set up the workpiece** function to set the preset:

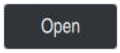
- Affix a real workpiece in the machine's working space

- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the workpiece at a notable point (e.g., a corner)

> This step makes the subsequent steps easier.

- Select the **Setup** application

- Select **Set up the workpiece**
- > The control opens the **Set up the workpiece** menu.
- Select NC program or 3D model



- Select **Open**
- > The control shows the 3D model in the simulation.

- If necessary, check the values of the active preset or choose a different preset

- Pre-position the 3D model by using the buttons for the individual axis directions within the virtual working space of the machine

> For pre-positioning the workpiece, use the workpiece touch probe as a point of reference.
>
> Even during the setup process, the shift functions are available for correcting the fixture position manually. Then, probe a new point.

- Specify the probing mode (e.g., **XY Clamping plane**)
- Position the workpiece touch probe until the control shows a green arrow pointing downward

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the workpiece will actually be probed. Check if the workpiece position in the simulation and in the machine match each other and if probing in the direction of the arrow is possible on the machine.
>
> Do not probe directly near edges, chamfers and roundings.

- Press the **NC Start** key
- > The control probes in the direction of the arrow.
- > The control displays the status of the **Z** axis in green and shifts the workpiece to the probed position. The control marks the probed position with a point in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
- > The control changes the status of the axes to green.

- Probe another point in axis direction **Y+** for the basic rotation
- > The control changes the status of the **SPC** spatial angle to green.
- Probe the check point in axis direction **X-**

Compensate the active preset

- Select **Compensate the active preset**
- > The control saves the determined values in the active row of the preset table.

←

- Exit the **Set up the workpiece** function

Note

NOTICE

Danger of collision!

To probe the clamping situation in the machine exactly, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision.

- Calibrate the workpiece touch probe at regular intervals
- Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the workpiece.
- Collisions might be more easily detected, if a tool carrier is assigned to the workpiece touch probe.
- HEIDENHAIN recommends probing check points for one axis direction on both sides of the workpiece. As a result, the control will correct the position of the 3D model in the simulation uniformly.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

17.4 Measuring the tool by scratching

Application

Not all machines are equipped with a tool touch probe for measuring a tool. The **Tool measured** touch probe function enables determining the tool dimensions by scratching a workpiece.

Related topics

- Touch probe functions in the **Setup** application
  Further information: "Touch probe functions in the Manual operating mode", Page 373
- Measuring the tool automatically with cycles
  Further information: Measuring Cycles for Workpieces and Tools User's Manual

Description of function

You do not use a 3D touch probe for scratching, but the tool to be measured. In the scratching process, approach the tool carefully to a workpiece surface until you can see a thin chip being removed. The handwheel allows obtaining a higher accuracy.

In the **X** or **Y** probing directions, the tool radius can be determined. When selecting probing direction **Z**, the tool length is measured.

Buttons in the Measure the tool function

The control offers the following options for writing the measured radius or length values into the tool table:

Button	Meaning
Write basic values	The control transfers the values into columns **R** or **L**. The control resets existing delta values in columns **DR** or **DL**.
Write delta values	The control enters the delta values in columns **DR** or **DL**.

Further information: "Tool tables", Page 457

17.4.1 Tool measurement by scratching

The dimensions of an end mill can be determined by using the **Tool measured** function as follows:

- Select the **Manual** operating mode
- Set the workpiece preset if required

> Position the workpiece preset on the surfaces to be scratched in order to obtain a clear reference.

- Insert the tool to be measured
- Define the speed if required
- Start the tool spindle
- Select the **Setup** application

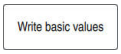
- Select the **Measure tool** probing function

- If necessary, check the values of the active preset or choose a different preset

- Scratch the workpiece in the desired axis direction (e.g., **X+**)

- Select the associated probing direction **X+**

- Select **actual position capture**
- > The control transfers the actual X axis position into the **Actual value** column.
- > The control shows the measurement results.
- Enter a **Nominal value** (e.g., **0**)



- Select **Write basic values**
- > The control transfers the value into column **R** of the tool table.
- > The control resets the existing delta value in the **DR** column.

> When selecting **Write delta values**, the control will enter only one delta value in column **DR**.

- Scratch another axis direction if required (e.g., **Z-**)

- Select **Exit probing**
- > The control closes the **Measure tool** probing function.

17.5 Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message remains active only for 30 seconds.

17.5.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:

- Select the **Manual** operating mode
- Select **Suppress touch probe monitoring**
- > The control disables touch-probe monitoring for 30 seconds.
- If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

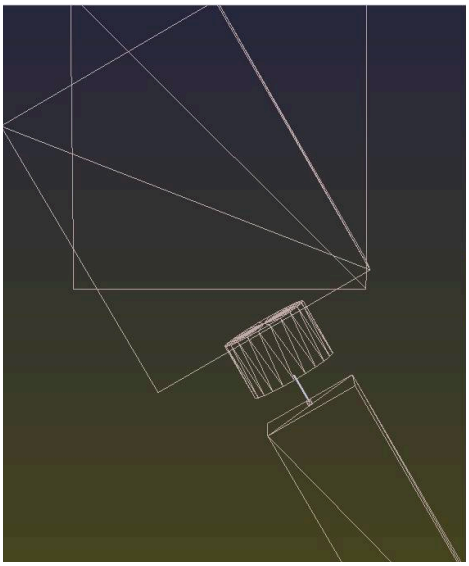
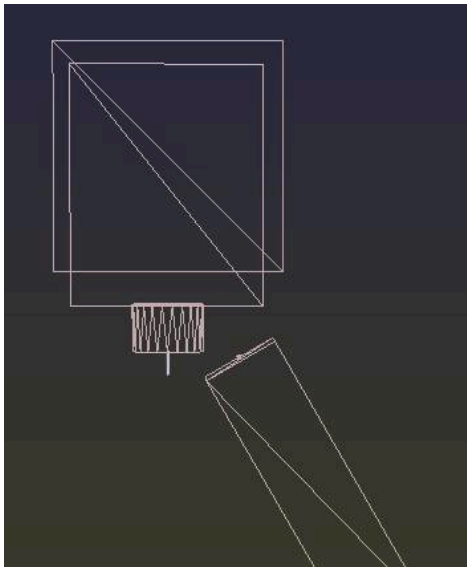
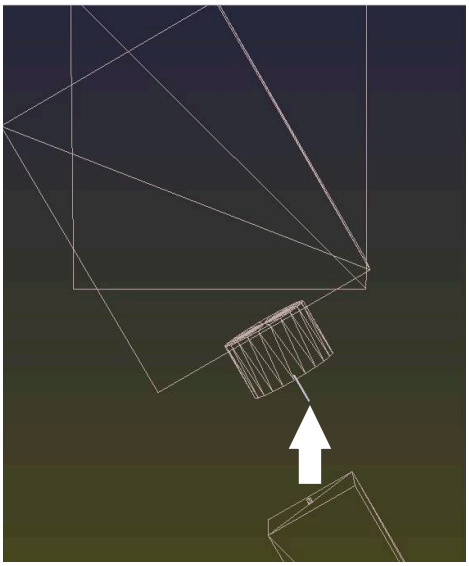
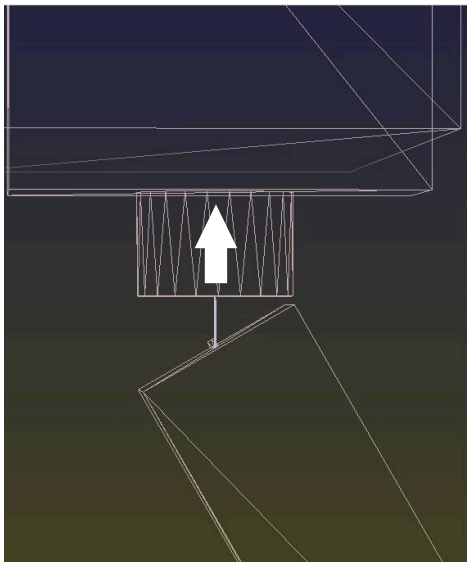
While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

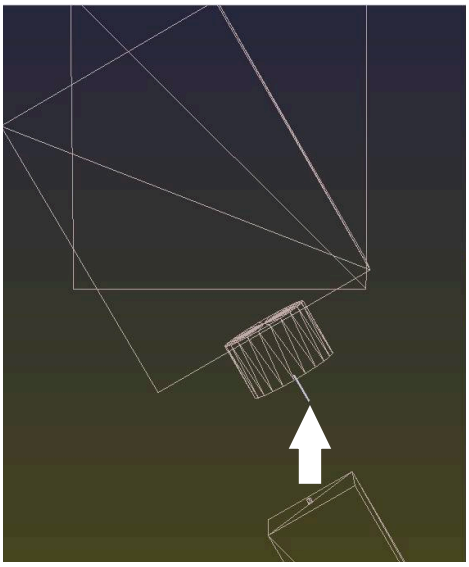
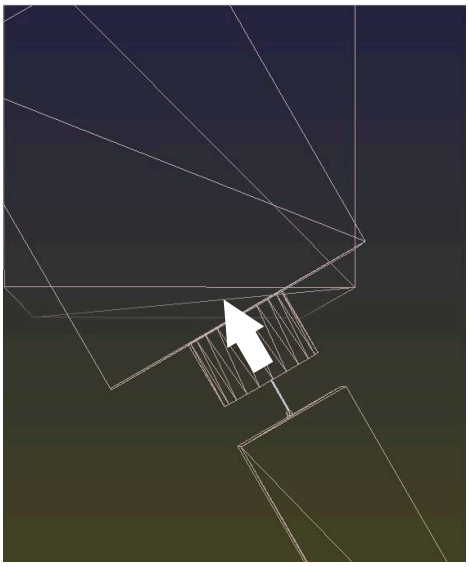
- Carefully move the axes in the **Manual** operating mode

If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

17.6 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

Offset	3D basic rotation
Initial state	Initial state
	
Position display: ■ Actual position ■ **B** = 0 ■ **C** = 0 Preset table: ■ **SPB** = 0 ■ **B_OFFS** = -30 ■ **C_OFFS** = +0	Position display: ■ Actual position ■ **B** = 0 ■ **C** = 0 Preset table: ■ **SPB** = -30 ■ **B_OFFS** = +0 ■ **C_OFFS** = +0
Movement in +Z without tilting	Movement in +Z without tilting
	

Offset	3D basic rotation
Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**	Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**
	
> The orientation **is not correct**!	> The orientation is correct! > The next machining step will be **correct**.

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

18

Program run

18.1 The Program Run operating mode

18.1.1 Fundamentals

Application

In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
  Further information: "The MDI Application ", Page 369
- Creating NC programs
  Further information: Programming and Testing User's Manual
- Pallet tables
  Further information: Programming and Testing User's Manual

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- ▶ Copy the NC program and all called files to the **TNC:** drive

NOTICE

Danger of collision!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. Undesirable machine movements or collisions may result.

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

Description of function

The following information also applies to pallet tables and job lists.

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 424

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In **Single Block** mode you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the TNC bar", Page 125

The **Program Run** operating mode provides the following workspaces:

- **GPS** (#44 / #1-06-1)
  Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287
- **Document**
  Further information: Programming and Testing User's Manual
- **Positions**
  Further information: "The Positions workspace", Page 119
- **Program**
  Further information: Programming and Testing User's Manual
- **Simulation**
  Further information: Programming and Testing User's Manual
- **Status**
  Further information: "The Status workspace", Page 127
- **Process Monitoring** (#168 / #5-01-1)
  Further information: Programming and Testing User's Manual

When opening a pallet table, the control displays the **Job list** workspace. You cannot modify this workspace.

Further information: Programming and Testing User's Manual

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
	**Open File** With **Open File** you can open a file (for example, an NC program). When you open a file, the control closes the file that was already open.
	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Single Block	If this toggle switch is active, then you run each NC block separately with the **NC Start** key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** Programming and Testing User's Manual
Compensation tables	The control opens a selection menu with the following tables: ■ **D** ■ **T-CS** ■ **WPL-CS** **Further information:** "Compensation during program run", Page 436
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 417
Automatic program start	Starts machining at a defined time automatically **Further information:** "Automatic program start", Page 433

Icon or button	Meaning
Program run options	When you select this button, the control opens the **Program run options** window with the following selection possibilities: ■ Settings for the override controller **Further information:** "The Program run options window", Page 540 ■ **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:** "The Program run options window", Page 540 ■ **Feed F LIMIT** Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 417 ■ **Skip block** If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual ■ **Pause at M1** If the toggle switch is active, the control stops the execution at every NC block with **M1**. If the toggle switch is inactive, then the control dims the **M1** syntax element. **Further information:** Programming and Testing User's Manual
GOTO Cursor	The control marks the table row currently selected for execution. This button is available when a pallet table is open. **Further information:** Programming and Testing User's Manual
AFC	Use this option to activate or deactivate Adaptive Feed Control (AFC (#45 / #2-31-1)). **Further information:** "The AFC toggle switch in the Program Run operating mode", Page 280
AFC settings	The control opens a selection menu with the following selection possibilities for AFC (#45 / #2-31-1): ■ **AFC.TAB** for AFC basic settings ■ **AFC.DEP** settings file for teach-in cuts of the active NC program ■ **AFC2.DEP** log file of the active NC program ■ **Stop Teach** **Further information:** "The AFC settings button", Page 283

Icon or button	Meaning
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a **/** character. If the toggle switch is active, then the control dims the NC blocks to be skipped. **Further information:** Programming and Testing User's Manual
Pause at M1	If the toggle switch is active, the control stops the execution at every NC block with **M1**. If the toggle switch is inactive, then the control dims the **M1** syntax element. **Further information:** Programming and Testing User's Manual
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC (#145 / #2-30-1)). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 286
Edit	If this toggle switch is active, then you can edit the pallet table. This button is available if a pallet table is open. **Further information:** Programming and Testing User's Manual
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** Programming and Testing User's Manual
Manual traverse	While a program run is interrupted, you can move the axes manually. If **Manual traverse** is active, the operating mode's icon in the control bar changes. **Further information:** "Manual traverse during an interruption", Page 423
3D ROT	While a program run is interrupted, you can move the axes manually in the tilted working plane (#8 / #1-01-1). **Further information:** "Manual traverse during an interruption", Page 423
Approach position	Return to contour after manual traverse of the machine axes during an interruption **Further information:** "Returning to the contour", Page 432
Block scan	Use the **Block scan** function to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with **M3**. **Further information:** "Block scan for mid-program startup", Page 424
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. **Further information:** "The Retract application", Page 438
Open in the editor	The control opens the active NC program in the **Editor** operating mode and selects the currently selected NC block, even for called NC programs. This button is available when an NC program is open. **Further information:** Programming and Testing User's Manual
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 197
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.

Icon or button	Meaning
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally active program information as well as the program run-time.

Feed rate limit F LIMIT

The **F LIMIT** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **F LIMIT** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **F LIMIT** button in the function bar, the control will open the **Feed rate F LIMIT** window.

Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

If a feed rate limit is active, the control highlights the **F LIMIT** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the feed rate is displayed in orange.

Further information: "Status displays", Page 117

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate F LIMIT** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M0**)
- Stop program run (e.g., with the **NC Stop** key)
- Cancel program run (e.g., with the **NC stop** key and the **Internal stop** button)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 365

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview on the TNC bar", Page 125

Below are some of the functions you can execute in an interrupted or canceled state:

- Selecting an operating mode
- Manual traverse of axes
- Checking variables and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the setting for the programmed skipping of NC blocks with **/**

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the **Q parameter list** window
  - Switching the operating modes
- Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

After canceling the program run with an **Internal stop**, you must start the program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of the program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 424

Modally effective program information

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour (**Approach position** button).

Further information: "Returning to the contour", Page 432

The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE

Danger of collision!

Program cancellation, manual intervention, forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision.

- Rescind all programmed NC functions and transformations within the NC program
- Run a simulation before executing an NC program
- Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program
- Carefully prove-out the NC program in **Single Block** mode

- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
- When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece preset during a program run interruption, you must re-select the NC block to resume.

  Further information: "Block scan for mid-program startup", Page 424
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
- The settings in the **GPS** workspace have an effect on the program run, such as handwheel superimpositioning (#44 / #1-06-1).

  Further information: "Global Program Settings GPS (#44 / #1-06-1)", Page 287
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

Definitions

Abbreviation	Definition
GPS (global program settings)	Global program settings
ACC (active chatter control)	Active Chatter Control

18.1.2 Navigation path in the Program workspace

Application

If you execute an NC program or a pallet table, or if you test it in the opened **Simulation workspace, the control will display a navigation path in the file information bar of the Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

Related topics

- Program call
  Further information: Programming and Testing User's Manual
- The **Program** workspace
  Further information: Programming and Testing User's Manual
- The **Simulation** workspace
  Further information: Programming and Testing User's Manual
- Interrupted program run
  Further information: "Interrupting, stopping or canceling program run", Page 418

Requirement

- The **Program** and **Simulation** workspaces are both opened
  In the **Editor** operating mode you need both workspaces to use the function.

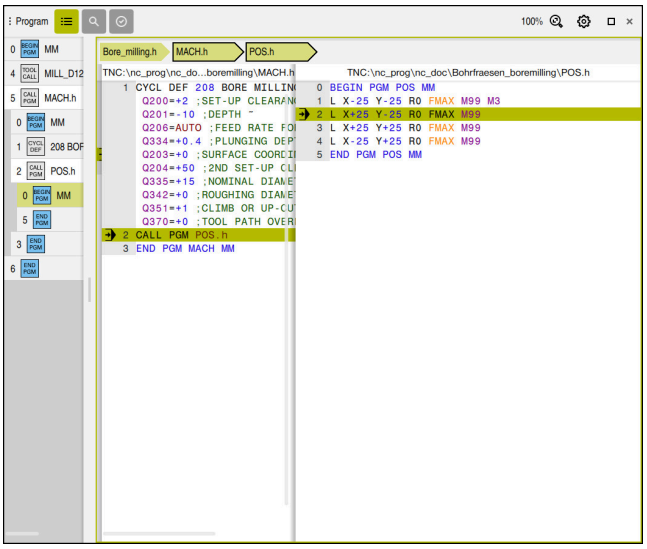
Description of function

The control shows the name of the NC program as a path element in the file information bar. As soon as the control calls a different NC program, the control adds a new path element with the name of the called NC program to the bar.

Additionally, the control displays the contents of the called NC program in a new pane in the **Program** workspace. The control displays as many NC programs side by side as the size of the workspace permits. If necessary, newly opened NC programs will cover previously opened NC programs. The control displays the covered NC programs in a narrow band at the left edge of the workspace.

When execution is interrupted, you can navigate between the NC programs. When you select the path element of an NC program, the control opens the content.

When you select the last path element, the control automatically marks the active NC block with the execution cursor. When you press the **NC Start** key, the control resumes execution of the NC program from this position.



Called NC programs in the **Program** workspace in the **Program Run** operating mode

Depiction of path elements

The control displays the path elements of the navigation path as follows:

Format	Meaning
Black frame	The NC program is visible in the **Program** workspace and is not covered by other NC programs.
Highlighted in green	The NC program at the current cursor position is active or is considered for program run. If, for example, the cursor is positioned in the called NC program, the calling NC program will be considered for program run.
Highlighted in gray	The NC program is active for execution but will not be considered for program run at the current cursor position. If, for example, you stop the execution and navigate into the calling NC program, the control displays the path element of the called NC program in gray.

Note

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.

The structure items allow you to navigate into every NC program. The control displays the associated NC programs in the **Program** workspace. The navigation path always remains at the current point of execution.

Further information: Programming and Testing User's Manual

18.1.3 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

The **Tilt the working plane (3D ROT)** window allows selecting the reference system in which you move the axes (#8 / #1-01-1).

Related topics

- Manual traverse of machine axes
  Further information: "Moving the machine axes", Page 164
- Tilting the working plane manually (#8 / #1-01-1)
  Further information: Programming and Testing User's Manual

Description of function

When you select **Manual traverse**, you can move the axes with the axis keys of the control.

Further information: "Using axis keys to move the axes ", Page 164

In the **Tilt the working plane (3D ROT)** window, you can select the following functions:

Icon	Function	Meaning
	**M-CS machine**	Traversing in the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216
	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 220
	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 222
	**T-CS tool**	Traversing in the tool coordinate system **T-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 222

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

If **Manual traverse** is active, then the operating mode's icon in the control bar changes.

Notes

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). Selecting an incorrect **3D ROT** setting or moving the tool in the wrong direction involves risk of collision!

- It is better to use the **T-CS** function
- Check the direction of movement
- Move at slow feed rate

- On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.
  Refer to your machine manual.

18.1.4 Block scan for mid-program startup

Application

The **Block scan** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs
  Further information: Programming and Testing User's Manual
- Pallet tables and job lists
  Further information: Programming and Testing User's Manual

Requirement

- The function must be enabled by your machine manufacturer.
  The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

The following information also applies to pallet tables and job lists.

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- The **Internal stop** button
- Emergency stop
- Power failure

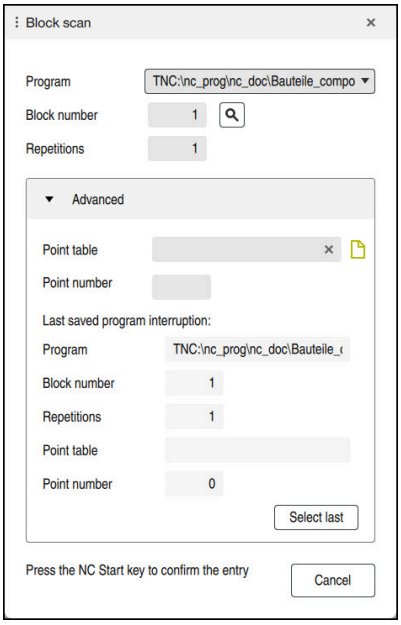
If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary
  Further information: "Performing a single-level block scan", Page 427
- Multi-level block scan in subprograms and touch probe cycles
  Further information: "Performing a multi-level block scan", Page 428
- Block scan in a point table
  Further information: "Block scan in point tables", Page 429
- Block scan in pallet programs
  Further information: "Block scan in pallet tables", Page 430

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During the block scan you can activate or deactivate **Single Block** mode.

The Block scan window



The **Block scan** window with saved interruption point and open **Advanced** area

The **Block scan** window provides the following data:

Row	Meaning
Row number	Row number in the pallet table
	Row number in the pallet table at the time of interruption
Program	Path of the active NC program
	Path of the NC program that is active at the time of interruption
Block number	Number of the NC block at which program run should start The control proposes the currently selected NC block. You can open the **Search** column from the **Block scan** window.
	Number of the NC block that was active at the time of interruption
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat or a program loop.
	Number of the repetition at the time of interruption if the interruption was located within a program-section repeat or program loop.
Point table	Path of the point table Selection by means of a selection window
	Path of the active point table at the time of interruption
Point number	Row in the point table
	Active row in the point table at the time of interruption

Select the interruption point by using the **Select last** button in the **Advanced** area.

Performing a single-level block scan

To start in an NC program by using a single-level block scan:

- Select the **Program Run** operating mode

- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

- If applicable, open the **Extended** area

- If required, use **Select last** to start at a saved interruption point

- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 433

- Press the **NC Start** key
- > The control resumes execution of the NC program.

Performing a multi-level block scan

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this, you first go to the desired subprogram call and then continue the block scan. The same procedure is used for called NC programs.

To start in an NC program by using a multi-level block scan:

- ▶ Select the **Program Run** operating mode

- ▶ Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- ▶ Perform a block scan to the first start-up point:

  Further information: "Performing a single-level block scan", Page 427

- ▶ Activate the **Single Block** toggle switch as needed

- ▶ Press the **NC Start** key to execute individual NC blocks as needed

- ▶ Select **Continue block scan**
- ▶ Define the NC block for mid-program startup

- ▶ Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.

- ▶ Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

- ▶ Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 433

- ▶ Select **Continue block scan** again as needed
- ▶ Repeat the steps

- ▶ Press the **NC Start** key
- > The control resumes execution of the NC program.

Block scan in point tables

To start in a point table:

- Select the **Program Run** operating mode

- Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.

- Open the **Extended** area

- Under **Point table**, select the point table
- **Point number**: Select the row number of the point table for mid-program startup

- Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

- Press the **NC Start** key
- Using the displayed positioning logic, the control moves to the required positions.

You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 433

If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired point for mid-program startup in the **Point number** field. The first point in the point pattern has the number 0.

Further information: User's Manual for Machining Cycles

Block scan in pallet tables

To start in a pallet table:

- Select the **Program Run** operating mode

- Select **Block scan**
- The control opens the **Block scan** window.
- **Row number**: Enter the row number of the pallet table
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

- If applicable, open the **Extended** area

- If required, use **Select last** to start at a saved interruption point

- Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

- Press the **NC Start** key
- Using the displayed positioning logic, the control moves to the required positions.

You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 433

If the program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Caution: Danger to the tool and workpiece!

If program run is interrupted within a called NC program, the control always offers the first call of this NC program as a point of interruption for mid-program startup. If program run was interrupted in a later call, the control might then execute parts of the program that have already run.

- Use multi-stage mid-program startup to manually navigate to the point of interruption

NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- Use the **Block scan** function in multiple steps

- The control only displays the dialogs required by the process in the pop-up window.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- If you use the block scan to start in a pallet table, the control will always execute the chosen row in the pallet table as a workpiece-oriented process. After the pallet table line selected in the **Block scan**, the control resumes machining according to the defined machining method.

  Further information: Programming and Testing User's Manual
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.

  Further information: "The LBL tab", Page 134
- The **Block scan** function must not be used in conjunction with the following functions:
  - Touch probe cycles **0**, **1**, **3**, and **4** during the block scan search phase
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.

18.1.5 Returning to the contour

Application

With the **RESTORE POSITION** function, the control moves the tool to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the **INTERNAL STOP** function.
- Return to the contour after a block scan (e.g., after an interruption with **INTERNAL STOP**)
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption

Related topics

- Manual traverse during program run interruptions

  Further information: "Manual traverse during an interruption", Page 423
- The **Block scan** function

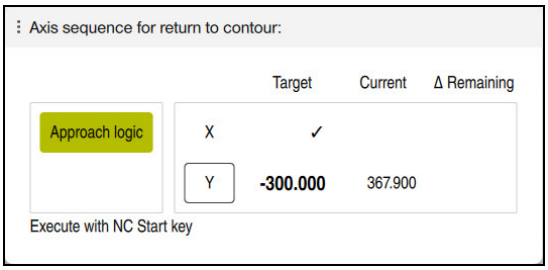
  Further information: "Block scan for mid-program startup", Page 424

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

The Axis sequence for return to contour: window



The **Axis sequence for return to contour:** window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 433

If manual axes are included in the axes to be returned to the contour, then the control will not suggest a positioning logic. As soon as you have correctly positioned the manual axis, the control will suggest a positioning logic for the remaining axes.

Further information: "Positioning manual axes", Page 433

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the desired axis (e.g., **X**)

- ▶ Press the **NC Start** key
- > The control moves the axis to the required position.
- > When the axis has reached the correct position, the control shows a checkmark under **Target**.
- ▶ Position the remaining axes
- > When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the manual axis (e.g., **W**)
- ▶ Position the manual axis to the value shown in the window
- > When a manual axis with encoder has reached the position, the control automatically clears the value.
- ▶ Select **Axis in position**
- > The control saves the position.

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

18.1.6 Automatic program start

Application

With the **Automatic program start** function you define a specific time at which the control will start to execute an NC program, such as a warm-up program for the machine. No machine operator needs to be present.

Requirements

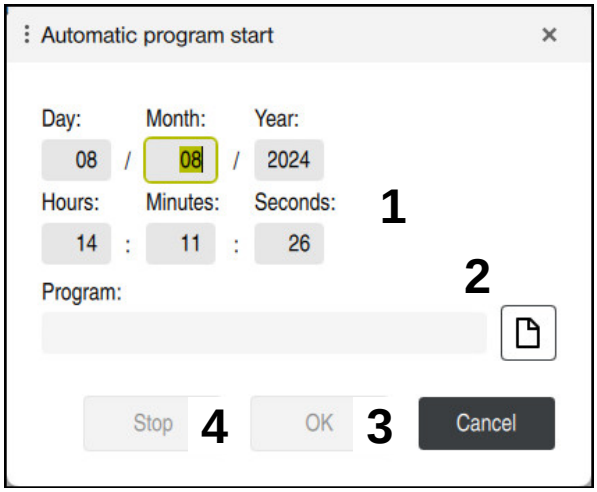
- The function must be enabled by your machine manufacturer.
  In the machine parameter **autoStartEnabled** (no. 100701), the machine manufacturer defines whether an automatic program start is possible.
- Machine is switched on
- No NC program is currently running on the machine
  If a program is already running at the specified time, the control will not execute an automatic program start.
- Machine is prepared for the NC program (e.g., correct workpiece preset is active)

> HEIDENHAIN recommends activating the workpiece preset in the NC program.
> **Further information:** Programming and Testing User's Manual

Description of function

Use this function to execute NC programs and pallet tables.

If you select the **Automatic program start** button, the control opens the **Automatic program start** window.



The **Automatic program start** window

The control offers the following possibilities in the **Automatic program start** window:

1 Defining the date and time
2 Selecting an NC program via a selection window
3 Activating automatic program start
4 Deactivating automatic program start

If an automatic program start is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 97

Notes

⚠ WARNING

Caution: hazard to the user!

If automatic program start is active, the machine starts an NC program on its own and moves the axes. There is an increased risk of injury on machines without housing.

- Clearly indicate on the machine whether **Automatic program start** is active
- If necessary, restrict access to the machine

NOTICE

Caution: Significant property damage!

If automatic program start is active, the machine starts an NC program without a machinist and moves the axes. Collisions can occur if the machine was not set up correctly, or if there have been changes in the meantime. There might be no machinist present in order to abort program run.

- Set up the machine appropriately for such machining, e.g.:
  - Install the correct workholding equipment
  - Measure the tools used in the NC program and stock them in the magazine
- Optimize the NC program for automation, e.g.:
  - Automatic presetting
  - Activating the most recently set preset
  - Calling the correct workholding equipment

- In the machine parameter **closeDialogOnOK** (no. 100702), the machine manufacturer defines whether the control closes the **Automatic program start** window after activation.
- In the machine parameter **useLastStartData** (no. 100703), the machine manufacturer defines whether the control offers the last defined settings when opening the **Automatic program start** window.
- Ensure that the tools being used have sufficient service life remaining. If necessary, define a replacement tool.

18.2 Compensation during program run

Application

During program run, you can open the selected compensation tables and the active datum table, and edit the values.

Related topics

- Using compensation tables
  Further information: Programming and Testing User's Manual
- Editing compensation tables in the NC program
  Further information: Programming and Testing User's Manual
- Contents and creation of compensation tables
  Further information: Programming and Testing User's Manual
- Contents and creation of a datum table
  Further information: Programming and Testing User's Manual
- Activating a datum table in the NC program
  Further information: Programming and Testing User's Manual

Description of function

The control opens the selected tables in the **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

18.2.1 Opening tables from within the Program Run operating mode

To open the compensation tables from within the **Program Run** operating mode:

- ▶ Select **Compensation tables**
- > The control displays a selection menu.
- ▶ Select the desired table
  - **D**: Datum table
  - **T-CS**: Compensation table ***.tco**
  - **WPL-CS**: Compensation table ***.wco**
- > The control opens the selected table in the **Tables** operating mode.

Notes

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

- When opening a table in the **Program Run** operating mode, the control will display the **M** status in the table tab. This status means that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

  Further information: "Status overview on the TNC bar", Page 125

18.3 The Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

You can also retract a tool when the working plane is tilted or retract an inclined tool.

Requirement

- This application must be enabled by your machine manufacturer.

  The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** toggle switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**

  Further information: "The Retract workspace", Page 439
- **Positions**

  Further information: "The Positions workspace", Page 119
- **Status**

  Further information: "The Status workspace", Page 127

The **Retract** application provides the following buttons in the function bar:

Button	Meaning
Retract	Retract the tool with the axis keys or the electronic handwheel
End retraction	Close the **Retract** application The control opens the **End retraction?** window and prompts you to answer a confirmation request.
Start values	Reset the entries in the **A**, **B**, **C**, and **Thread pitch** fields to their original values

You select the **Retract** application by using the **Retract** toggle switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- The **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit F LIMIT", Page 417

The Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ **Machine axes**: Move in the machine coordinate system **M-CS** ■ **Tilted system**: Move in the working plane coordinate system **WPL-CS** (#8 / #1-01-1) ■ **Thread**: Move in the tool coordinate system **T-CS** with compensating movements of the spindle **Further information:** "Reference systems", Page 214
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes Effective in the **Tilted system** traverse mode
Thread pitch	Thread pitch from the **PITCH** column of tool management Effective in the **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ **Right-hand thread** ■ **Left-hand thread** Effective in the **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective in the **Tool axis** traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

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

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

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace

- ▶ Activate the **Retract** toggle switch

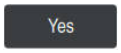
- ▶ Press **OK**
- > The control compiles the PLC program.

- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- ▶ Compare the displayed position values with the actual position values

- ▶ Select **OK**
- > The control closes the **Assume position values?** window
- ▶ Select the **Thread** traverse mode as needed
- ▶ Enter the thread pitch as needed
- ▶ Enter the direction of rotation as needed

- ▶ Select **Retract**
- ▶ Retract the tool with the axis keys or the handwheel

- ▶ Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.



- ▶ If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

19

Tables

19.1 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: Programming and Testing User's Manual

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: Programming and Testing User's Manual

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 447

Further information: "The Form workspace for tables", Page 454

You can select various functions by using the context menu (e.g., **Copy**).

Further information: Programming and Testing User's Manual

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:** **Further information:** "Tool types", Page 194 ■ **Line number (Tool number?)** ■ **Number of rows** ■ **Index** **Further information:** "Indexed tool", Page 188 ■ **Append** Append rows at the end of the table **Further information:** "Tool management ", Page 197
Delete tool	The control deletes the tool selected in the tool management. You cannot delete any tools that have been entered into the pocket table. The button is dimmed. **Further information:** "Tool management ", Page 197
Import	The control imports tool data. **Further information:** "Importing tool data", Page 199
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.
Activate the preset	The control activates the currently selected row of the preset table as preset. **Further information:** "Preset table *.pr", Page 503
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes. **Further information:** "Write-protection for table rows", Page 507

Refer to your machine manual.
If necessary, the machine manufacturer adapts the buttons.

19.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Enable **Edit**
- > The control enables the values for editing.

> 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.

> 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.

  > 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.

19.2 The Create new table window

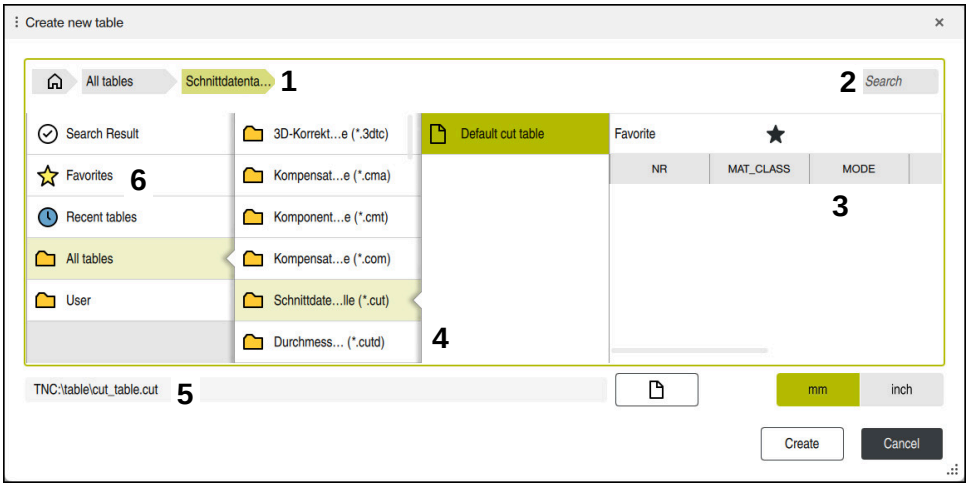
Application

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: Programming and Testing User's Manual
- Available file types for tables
  Further information: Programming and Testing User's Manual

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

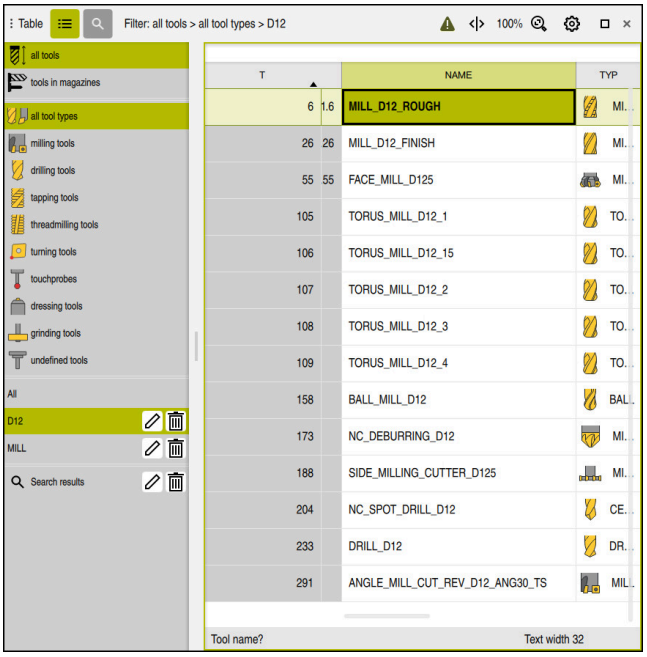
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer 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**).

19.3 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 449
 **CTRL + F**	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 452
	■ 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 **CfgTableCellCheck** (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:** Programming and Testing User's Manual
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 452
	Open the **Search** column and edit the selected filter Only in the **Filter** column **Further information:** "The Search column in the Table workspace", Page 452
	Delete the selected filter Only in the **Filter** column **Further information:** " Filter column in the Table workspace", Page 449
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

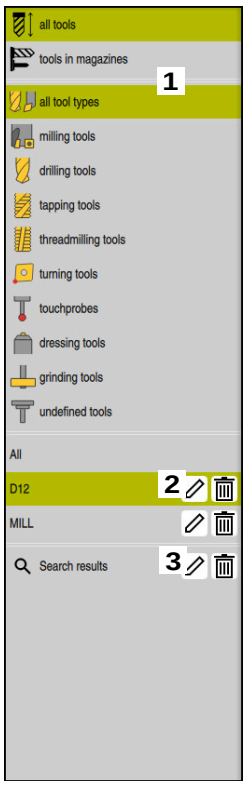
The control offers default filters for the following tables:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

For all other tables, you can create user-defined filters.

Further information: "User-defined filters", Page 450

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 450

3 **Search results**

Results of the **Search** column

Further information: "The Search column in the Table workspace", Page 452

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 451

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 452

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 449

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

> 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 450

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 449

> 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:

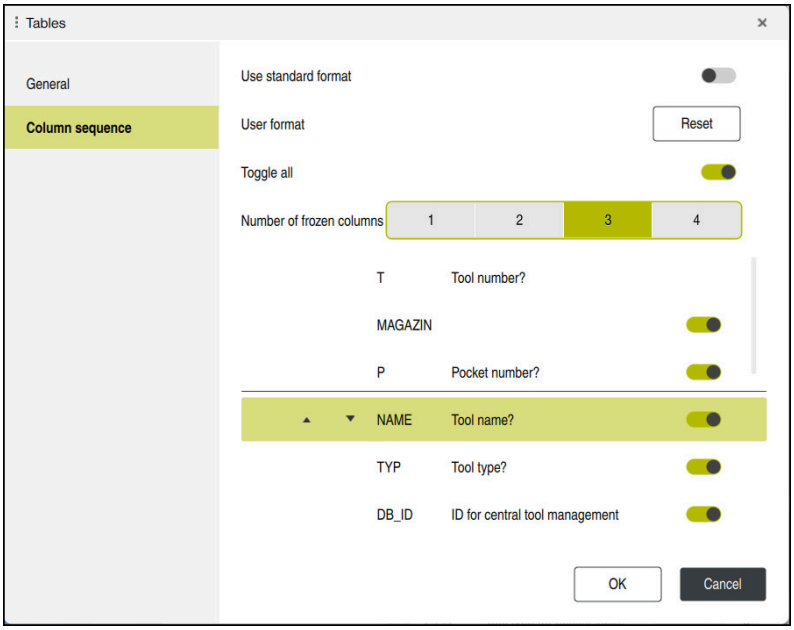
- **General**
- **Column sequence**

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.

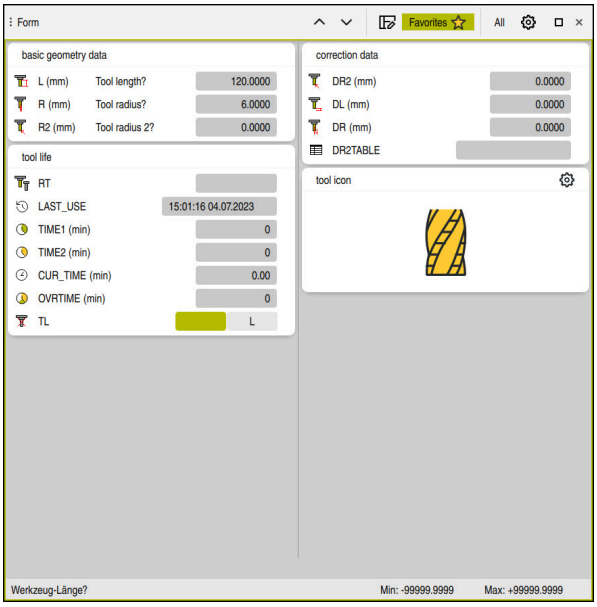
The settings in the **Column sequence** area only apply to the currently opened table.

19.4 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).

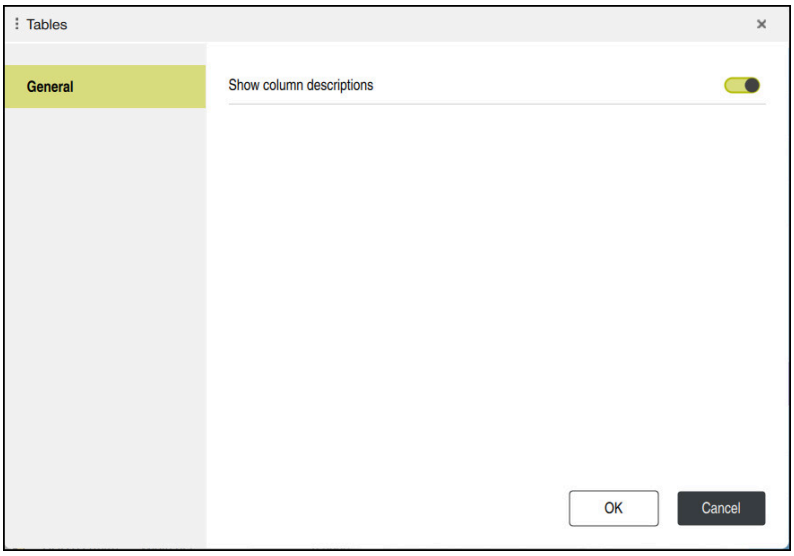
Buttons and icons

The **Form** workspace contains the following buttons, icons or shortcuts:

Buttons, icons or shortcuts	Meaning
^ SHIFT + UP v SHIFT + DOWN	**Navigate** Navigate between table rows
	**Configure the layout** You can make the following layout adaptations: ■ Add or remove areas to the **Favorites** view ■ Rearrange areas using the gripper ■ Add or remove columns
Favorites	In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites.
All	In this view the control shows all areas.
	**Settings** ■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 456 ■ 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 456 ■ 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.



19.4.1 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: "Tool types", Page 194
- The control displays help graphics on how the parameters for grinding tools will be in effect (#156 / #4-04-1).

  Further information: Programming and Testing User's Manual

19.5 Tool tables

19.5.1 Overview

This chapter contains information about the tool tables of the control.

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 457
- Turning tool table **toolturn.trn** (#50 / #4-03-1)
  Further information: "Turning tool table toolturn.trn (#50 / #4-03-1)", Page 468
- Grinding tool table **toolgrind.grd** (#156 / #4-04-1)
  Further information: "Grinding tool table toolgrind.grd (#156 / #4-04-1)", Page 473
- Dressing tool table **tooldress.drs** (#156 / #4-04-1)
  Further information: "Dressing tool table tooldress.drs (#156 / #4-04-1)", Page 485
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 489

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 197

19.5.2 Tool table tool.t

Application

The tool table **tool.t** contains the parameters specific to drilling and milling tools. The tool table also contains all parameters that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool parameters
  Further information: "Tool parameters", Page 185

Description of function

The file name of the tool table is **tool.t** and this table must be stored in the folder **TNC:\table**.

Parameters of the tool table tool.t

The **tool.t** tool table provides the following parameters:

Parameter	Meaning
T	**Tool number?** Row number in the tool table The tool number allows you to identify each tool unambiguously (e.g., for calling a tool). **Further information:** Programming and Testing User's Manual You can define an index after the period. **Further information:** "Indexed tool", Page 188 This parameter applies to all tools, regardless of technology. Input: **0.0...32767.9**

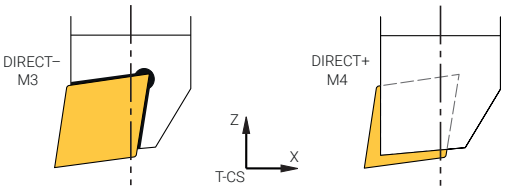
Parameter	Meaning
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** Programming and Testing User's Manual You can define an index after a period (i.e., name.index). **Further information:** "Indexed tool", Page 188 This parameter applies to all tools, regardless of technology. Input: **Text width 32**
L 	**Tool length?** Length of tool, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
R 	**Tool radius?** Tool radius, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
R2 	**Tool radius 2?** Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of, for example, ball-nose cutters or toroid cutters. **Further information:** Programming and Testing User's Manual Input: **-99999.9999...+99999.9999**
DL 	**Tool length oversize?** Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter **L** Input: **-999.9999...+999.9999**
DR 	**Tool radius oversize?** Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to parameter **R** Input: **-999.9999...+999.9999**
DR2 	**Tool radius oversize 2?** Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to parameter **R2** Input: **-999.9999...+999.9999**

Parameter	Meaning
TL 	**Tool locked?** Tool is enabled or locked for machining: ■ No value entered: Enabled ■ **L**: Locked The control locks the tool after exceeding maximum tool age **TIME1**, maximum tool age 2 **TIME2** or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: No value, **L**
RT 	**Replacement tool?** Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool at a suitable location. **Further information:** Programming and Testing User's Manual If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool. You can define an index after the period. **Further information:** "Indexed tool", Page 188 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **0.0...32767.9**
TIME1 	**Maximum tool age?** Maximum tool age in minutes If the current tool age **CUR_TIME** exceeds the **TIME1** value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**

Parameter	Meaning
TIME2	**Max. tool age for TOOL CALL?** Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: ■ When the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool **RT** is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool **RT** at a suitable location. **Further information:** Programming and Testing User's Manual The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**
CUR_TIME	**Current tool age?** The current tool age equals the time during which the tool is cutting a workpiece. The tool is cutting a workpiece when the spindle is switched on and the control moves the tool at the machining feed rate. The control counts this time automatically and enters the current tool age in minutes. You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring. The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program. This parameter applies to all tools, regardless of technology. Input: **0...99999.99**
TYP	**Tool type?** Depending on the selected tool type, the control displays the suitable parameters in the **Form** workspace of the tool management. **Further information:** "Tool types", Page 194 **Further information:** "Tool management ", Page 197 This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **MILL**, **MILL_R**, **MILL_F**, **MILL_FACE**, **BALL**, **TORUS**, **MILL_CHAMFER**, **DRILL**, **TAP**, **CENT**, **TURN**, **TCHP**, **REAM**, **CSINK**, **TSINK BOR**, **BCKBOR**, **GF**, **GSF**, **EP**, **WSP**, **BGF**, **ZBGF**, **GRIND**, and **DRESS**
DB_ID	**ID for central tool management** The database-ID allows you to identify a tool (e.g., within a tool management system by using client applications). **Further information:** "Database ID", Page 187 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool. **Further information:** "Indexed tool", Page 188 This parameter applies to all tools, regardless of technology. Input: **Text width 40**

Parameter	Meaning
DOC	**Tool description** This parameter applies to all tools, regardless of technology. Input: **Text width 32**
PLC	**PLC status?** Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology. Entry: **%00000000...%11111111**
LCUTS 	**Tooth length in the tool axis?** Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring. Input: **-99999.9999...+99999.9999**
LU 	**Usable length of the tool?** Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., of necks of end mills). Input: **0...999.9999**
RN 	**Neck radius of the tool?** Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of, for example, necks of end mills or side milling cutters. The tool can contain a neck radius **RN** only if the useful length **LU** is longer than the **LCUTS** length of the cutting edge. Input: **0...999.9999**
R_TIP 	**Radius at the tip** Radius at the tool tip for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of tools such as countersinks. Input: **0...999.9999**
ANGLE 	**Maximum plunge angle?** Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles. Input: **-360.00...+360.00**
CUT 	**Number of teeth?** Number of teeth of the tool for automatic tool measurement or cutting data calculation. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual **Further information:** Programming and Testing User's Manual This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0...99**

Parameter	Meaning
TMAT 	**Tool material?** Tool material from the tool material table **TMAT.tab** for cutting data calculation. **Further information:** Programming and Testing User's Manual Selection by means of a selection window Input: **Text width 32**
CUTDATA 	**Cutting data table?** Select the cutting data table with the ***.cut** or ***.cutd** file extension for cutting data calculation. **Further information:** Programming and Testing User's Manual Selection by means of a selection window Entry: **Text width 20**
LTOL 	**Wear tolerance: length?** Permitted tool length deviation in wear detection for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the **TL** parameter. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0.0000...5.0000**
RTOL 	**Wear tolerance: radius?** Permitted tool radius deviation in wear detection for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the **TL** parameter. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0.0000...5.0000**
R2TOL	**Wear tolerance: Radius 2?** Permitted tool radius 2 deviation in wear detection for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the **TL** parameter. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0...9.9999**

Parameter	Meaning
DIRECT  	**Cutting direction?** Cutting direction for exact definition of a tool for graphical representation, automatic tool measurement, and calculation of traverse movements. For milling tools you indicate the rotational direction of the tool spindle with which the tool cuts. ■ **-**: **M3** ■ **+**: **M4** For turning tools you indicate the rotational direction of the tool spindle when the tool is in front of the workpiece. Orient the tool and view it in the direction of **Y+** in the **T-CS** tool coordinate system: ■ **-**: You see the cutting edge on the front of the tool. The tool must move toward you in order to cut, meaning in the direction **Y-** (**M3**). ■ **+**: The cutting edge is on the rear of the tool. The tool must move away from you in order to cut, meaning in the direction **Y+** (**M4**).  Input: **-**, **+**
R-OFFS 	**Tool offset: radius?** Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **-99999.9999...+99999.9999**
L-OFFS 	**Tool offset: length?** Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the machine parameter **offsetToolAxis** (no. 122707) This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **-99999.9999...+99999.9999**

Parameter	Meaning
LBREAK 	**Breakage tolerance: length?** Permitted tool length deviation in breakage detection for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the **TL** parameter. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0.0000...9.0000**
RBREAK 	**Breakage tolerance: radius?** Permitted tool radius deviation in breakage detection for automatic tool measurement. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the **TL** parameter. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (#50 / #4-03-1) Input: **0.0000...9.0000**
NMAX 	**Maximum speed [rpm]** Limitation of spindle speed for the programmed value including control by the potentiometer. Input: **0...999999**
LIFTOFF 	**Lift-off allowed?** Allow automatic tool lift-off with active **M148** or **FUNCTION LIFTOFF**: ■ **Y**: Activate **LIFTOFF** ■ **N**: Deactivate **LIFTOFF** **Further information:** Programming and Testing User's Manual Selection by means of a selection window Input: **Y**, **N**
TP_NO	**Number of the touch probe** Number of touch probe in the touch probe table **tchprobe.tp** **Further information:** "Touch probe table tchprobe.tp", Page 489 Input: **0...99**
T-ANGLE 	**Point angle** Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of drilling tools, for example. **Further information:** User's Manual for Machining Cycles Input: **-180...+180**
PITCH 	**Tool thread pitch?** Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. **Further information:** User's Manual for Machining Cycles Input: **-9.9999...+9.9999**

Parameter	Meaning
AFC	**Feedback-control strategy** Control setting for adaptive feed control (AFC (#45 / #2-31-1)) from the **AFC.tab** table **Further information:** "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276 Selection by means of a selection window Entry: **Text width 10**
AFC-LOAD	**Reference power for AFC [%]** Tool-dependent reference power for AFC (#45 / #2-31-1). The input in percent refers to the rated spindle power. The control immediately uses the value given for feedback control, meaning a teach-in cut is dropped. Calculate the value beforehand with a teach-in step. **Further information:** "AFC teach-in cut", Page 282 Input: **1.0...100.0**
AFC-OVLD1	**AFC overload warning level [%]** Cut-related tool wear monitoring for AFC (#45 / #2-31-1). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Monitoring tool wear and tool load", Page 284 Input: **0.0...100.0**
AFC-OVLD2	**AFC ovrload switch-off level [%]** Cut-related tool load monitoring for AFC (#45 / #2-31-1). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. Is this parameter contains a value, the control ignores the **AFC-OVLD1** parameter. **Further information:** "Monitoring tool wear and tool load", Page 284 Input: **0.0...100.0**
LAST_USE 	**Date/time of last tool usage** The time at which the tool was last used The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program. This parameter applies to all tools, regardless of technology. Entry: **Text width 20**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table **Further information:** "Pocket table tool_p.tch", Page 494 Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
ACC	**ACC active?** Activate or deactivate active chatter control (ACC (#145 / #2-30-1)): ■ **Y**: Activate ■ **N**: Deactivate **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 286 Selection by means of a selection window Input: **Y**, **N**

Parameter	Meaning
KINEMATIC 	**Tool-carrier kinematics** Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring. **Further information:** "Tool carrier management", Page 201 Selection by means of a selection window This parameter applies to all tools, regardless of technology.
TSHAPE 	**3D tool model** Assigning a 3D model for exact definition of the tool for graphical simulation and collision monitoring. **Further information:** "Tool model (#140 / #5-03-2)", Page 205 Selection by means of a selection window
DR2TABLE	**Compensation val. table for DR2** Assigning a compensation value table ***.3dtc** for 3D tool radius compensation depending on the contact angle (#92 / #2-02-1). This allows the control to compensate for inaccuracies in the shape of a ball-nose cutter or the deflection behavior of a touch probe, for example. **Further information:** Programming and Testing User's Manual Selection by means of a selection window
OVRTIME 	**Tool life expired** Time in minutes during which the tool may be used beyond the tool life defined in the **TIME2** parameter. The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control will use the parameter when searching for tool names. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
RCUTS 	**Width of the indexable insert** Front-face width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., for indexable inserts). Input: **0...99999.9999**

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 493
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control changes only the position of the tool in the simulation.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    Refer to your machine manual.

    If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.
- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.

  The parameter **L-OFFS** is added to this defined distance.
- In the machine parameter **zeroCutToolMeasure** (no. 122724), the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.
- The machine manufacturer use the optional machine parameter **trackAsync** (no. 122503) to define whether the control orients the spindle during pre-positioning for probing. This can save time during automatic probing procedures. Additionally, the control takes the calibrated center offset of L-shaped styli into account for the spindle tracking speed. This means that the speed at the ball tip is at most the rapid traverse of the probe **FMAX**, which increases safety during probing.

19.5.3 Turning tool table toolturn.trn (#50 / #4-03-1)

Application

The turning tool table **toolturn.trn** contains the parameters specific to turning tools.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool parameters
  Further information: "Tool parameters", Page 185
- Milling-turning operations on the control
  Further information: Programming and Testing User's Manual
- General parameters, regardless of the technology
  Further information: "Tool table tool.t", Page 457

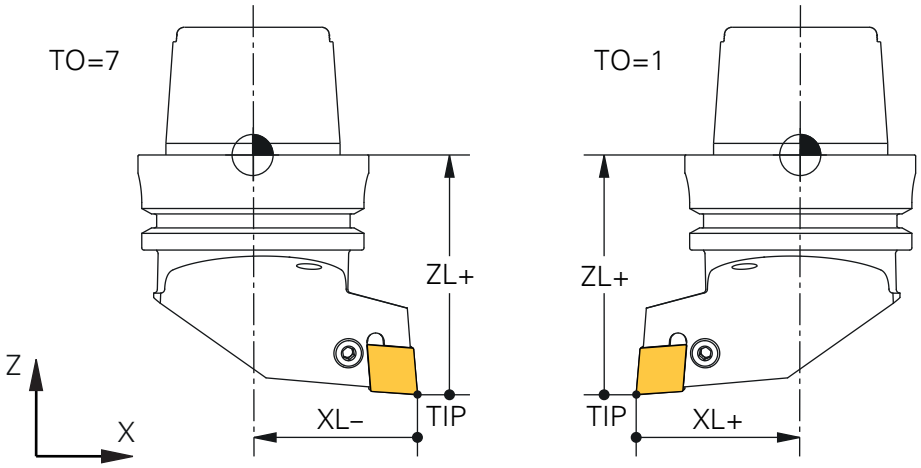
Requirements

- The Turning (#50 / #4-03-1) or Adv. Spindle Interpol. (#96 / #7-04-1) software option
- Turning tool is defined in **TYP** column of tool management
  Further information: "Tool types", Page 194

Description of function

The file name of the turning tool table is **toolturn.trn** and this table must be stored in the folder **TNC:\table**.

The values of the parameters **ZL**, **XL** and **YL** go from the tool tip TIP out to the tool-carrier reference point. The algebraic sign of, for example, **XL** depends on whether the tool tip is to the right or left of the tool spindle. If the tool is oriented and tool tip is to the right of the spindle center, enter a negative value for **XL**.



For **YL** the algebraic sign depends on whether the tool tip is in front of or behind the center of the tool spindle. If the tool tip is in front of the spindle center, enter a positive value for **YL**.

Parameters of the turning tool table toolturn.trn

The **toolturn.trn** turning tool table provides the following parameters:

Parameter	Meaning
T	Row number in the turning tool table The tool number allows you to identify each tool unambiguously (e.g., for calling a tool). **Further information:** Programming and Testing User's Manual You can define an index after the period. **Further information:** "Indexed tool", Page 188 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** Programming and Testing User's Manual You can define an index after a period (i.e., name.index). **Further information:** "Indexed tool", Page 188 Input: **Text width 32**
ZL 	**Tool length 1?** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2?** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3?** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
DZL 	**Oversize in tool length 1?** Delta value of tool length 1 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Oversize in tool length 2?** Delta value of tool length 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**

Parameter	Meaning
DYL 	**Tool length oversize 3?** Delta value of tool length 3 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Cutting edge radius?** The control takes into account the cutter radius for tool tip radius compensation. **Further information:** Programming and Testing User's Manual In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **TO**, **T-ANGLE**, and **P-ANGLE**. Input: **0...99999.9999**
DRS 	**Cutter radius oversize?** Delta value of cutter radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter **RS** Input: **-999.9999...+999.9999**
TO 	**Tool orientation?** From the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction. This information is necessary, for example, for calculating the cutter radius compensation, milling cutter radius compensation, plunge angle, etc. **Further information:** Programming and Testing User's Manual Refer to your machine manual. The control displays the tool orientations that are possible for each tool type. The machine manufacturer can change this assignment. In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **T-ANGLE**, and **P-ANGLE**. Input: **1...19**

Parameter	Meaning
ORI 	**Angle of spindle orientation?** Angle of tool spindle for aligning the turning tool Input: **-360.000...+360.000**
SPB-INSERT 	**Angular offset?** Angular offset for recessing and threading tools, spatial angle B Input: **-90.0...+90.0**
P-ANGLE 	**Point angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **T-ANGLE**. Input: **0...179.999**
T-ANGLE 	**Tool angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **P-ANGLE**. Input: **0...179.999**
CUTLENGTH  	**Cutting length of recessing tool** Usable length of the cutting edge of a turning or recessing tool. The control monitors the usable length of the cutting edge in the turning cycles. If the programmed cutting depth is greater than the usable length of the cutting edge defined in the tool table, then the control will display a warning and will automatically reduce the cutting depth. **Further information:** User's Manual for Machining Cycles If you do not define **CUTWIDTH**, the control uses the usable cutting length to define the tool for the graphic representation. The control calculates the missing information from the **CUTLENGTH**, **P-ANGLE** and **T-ANGLE** parameters. If the usable cutting length is less than the actual cutting length, the graphic representation will not match the actual tool. Input: **0...99999.9999**
CUTWIDTH  	**Width of recessing tool** Cutting width of a turning or recessing tool The control uses **CUTWIDTH** for calculations within cycles and to exactly define the tool for the graphic representation. **Further information:** User's Manual for Machining Cycles Input: **0...99999.9999**

Parameter	Meaning
DCW 	**Oversize f. recessing tool width** Delta value of recessing tool width as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** Measuring Cycles for Workpieces and Tools User's Manual Is added to parameter **CUTWIDTH** Input: **-99999.9999...+99999.9999**
TYPE 	**Type of turning tool** Depending on the selected turning tool type, the control displays the suitable parameters in the **Form** workspace of the tool management. **Further information:** "Turning tool types (#50 / #4-03-1)", Page 195 **Further information:** "Tool management ", Page 197 Selection by means of a selection window Input: **ROUGH**, **FINISH**, **THREAD**, **RECESS**, **BUTTON**, and **RECTURN**
WPL-DX-DIAM	**Compensation value for the workpiece diameter** Compensation value for the workpiece diameter with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 222 Input: **-99999.9999...+99999.9999**
WPL-DZL	**Compensation value for the workpiece length** Compensation value for the workpiece length with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 222 Input: **-99999.9999...+99999.9999**

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control changes only the position of the tool in the simulation.
- Geometry values from the tool table **tool.t**, such as length **L** or radius **R**, are not effective with turning tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    Refer to your machine manual.

    If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 493
- The columns **WPL-DX-DIAM** and **WPL-DZL** are deactivated in the default configuration.

  In the machine parameter **columnKeys** (no. 105501), the machine manufacturer activates the columns **WPL-DX-DIAM** and **WPL-DZL**. The names of the columns may be different, however.

19.5.4 Grinding tool table toolgrind.grd (#156 / #4-04-1)

Application

The grinding tool table **toolgrind.grd** contains the parameters specific to grinding tools.

Related topics

- Editing parameters in tool management

  Further information: "Tool management ", Page 197
- Tool parameters

  Further information: "Tool parameters", Page 185
- Grinding operations on milling machines

  Further information: Programming and Testing User's Manual
- Tool table for dressing tools

  Further information: "Dressing tool table tooldress.drs (#156 / #4-04-1)", Page 485
- General parameters, regardless of the technology

  Further information: "Tool table tool.t", Page 457

Requirements

- Grinding (#156 / #4-04-1) software option
- Grinding tool is defined in the **TYPE** column of tool management
  Further information: "Tool types", Page 194

Description of function

NOTICE

Danger of collision!

In the tool management form, the control displays only the parameters relevant to the selected tool type. The tool tables contain locked parameters that are for internal consideration only. If you edit these additional parameters manually, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

- Edit the tools in the tool management form

NOTICE

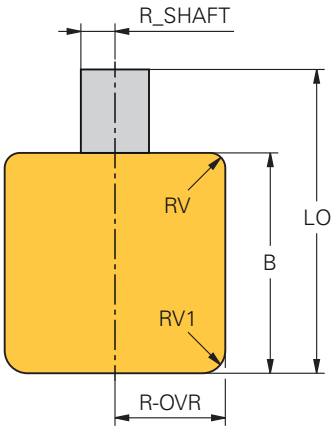
Danger of collision!

The control differentiates between freely editable and locked parameters. The control writes to the locked parameters and uses these parameters for internal consideration. You must not manipulate these parameters. If you manipulate the locked parameters, tool data might no longer correctly match each other. There is a risk of collisions during subsequent movements!

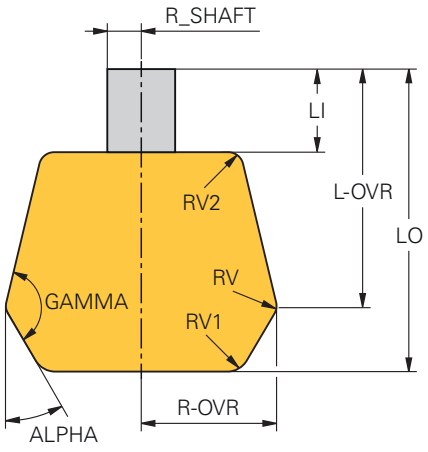
- Edit only freely editable tool management parameters
- Comply with the information about locked parameters in the tool data overview table

The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

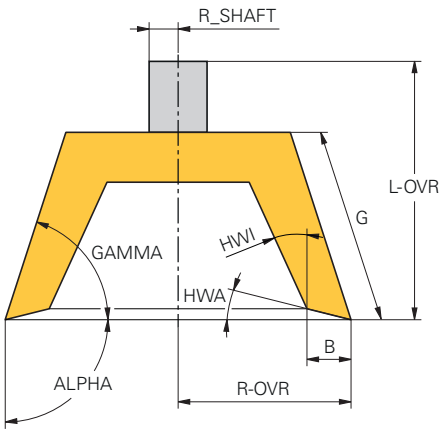
You define the parameters depending on the following grinding tool types:



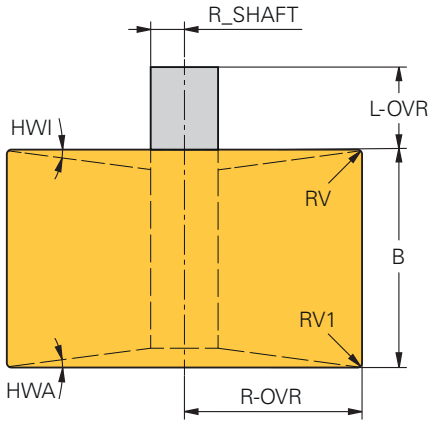
Cylindrical grinding pin



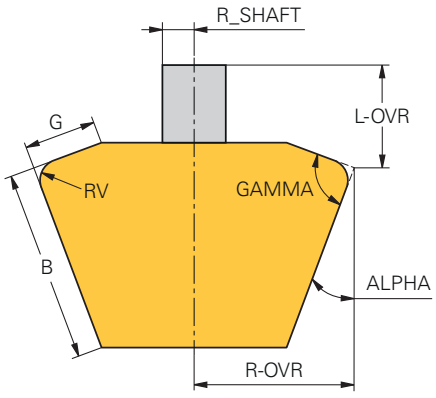
Conical grinding pin

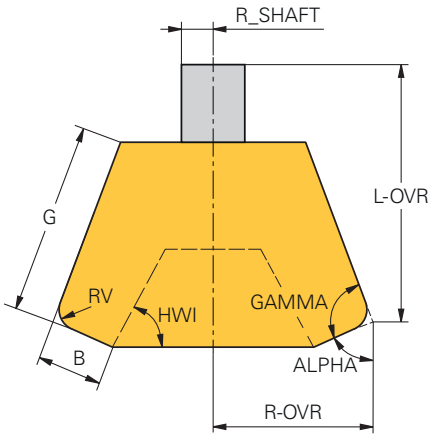


Cup wheel



Straight wheel and **Facing wheel**





Angular wheel

Parameters of the grinding tool table toolgrind.grd

The **toolgrind.grd** grinding tool table provides the following parameters:

Parameter	Meaning
T	**Tool number** Row number in the grinding tool table The tool number allows you to identify each tool unambiguously (e.g., for calling a tool). **Further information:** Programming and Testing User's Manual You can define an index after the period. **Further information:** "Indexed tool", Page 188 The row number must match the tool number in the **tool.t** tool table Input: **0...32767**
NAME	**Name of grinding wheel** The tool name identifies a tool, for example when calling it. **Further information:** Programming and Testing User's Manual You can define an index after a period (i.e., name.index). **Further information:** "Indexed tool", Page 188 Input: **Text width 32**
TYPE 	**Type of grinding wheel** Depending on the selected grinding tool type, the control displays the suitable parameters in the **Form** workspace of the tool management. **Further information:** "Grinding tool types (#156 / #4-04-1)", Page 196 **Further information:** "Tool management ", Page 197 Selection by means of a selection window Input: **GRIND_PIN**, **GRIND_CONE**, **GRIND_CUP**, **GRIND_CYLINDER**, **GRIND_ANGULAR** and **GRIND_FACE**
R-OVR 	**Radius of grinding wheel** Outermost radius of grinding tool After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
L-OVR 	**Overhang of grinding wheel** Length up to the outermost radius of the grinding tool, with respect to the tool carrier reference point After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
LO 	**Overall length** Absolute length of the grinding tool, with respect to the tool carrier reference point After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**

Parameter	Meaning
LI 	**Length to the inner edge** Length up to the inner edge, with respect to the tool carrier reference point After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
B 	**Width** Width of the grinding tool After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
G 	**Depth** Depth of grinding wheel After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
ALPHA	**Angle for the slant** After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.00000...90.00000**
GAMMA	**Angle for the corner** After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **45.00000...180.00000**
RV 	**Radius at the edge for L-OVR** **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
RV1 	**Radius at the edge for LO** **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
RV2 	**Radius at the edge for LI** **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
dR-OVR 	**Compensation of the radius** Delta value of the radius for tool compensation The control uses this parameter only if **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** has been selected in parameter **COR_TYPE**. The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **R-OVR** Input: **-999.999999...+999.999999**

Parameter	Meaning
dL-OVR 	**Compensation of the overhang** Delta value of the overhang for tool compensation The control uses this parameter only if **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** has been selected in parameter **COR_TYPE**. The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **L-OVR** Input: **-999.999999...+999.999999**
dLO 	**Compensation of the total length** Delta value of the total length for tool compensation The control uses this parameter only if **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** has been selected in parameter **COR_TYPE**. The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **LO** Input: **-999.999999...+999.999999**
dLI 	**Compensation of the length to the inner edge** Delta value of the length up to the inner edge for tool compensation The control uses this parameter only if **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** has been selected in parameter **COR_TYPE**. The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **LI** Input: **-999.999999...+999.999999**
R_SHAFT 	**Radius of the tool shank** Input: **0.00000...999.99999**
R_MIN 	**Min. permissible radius** If, after dressing, the actual radius is below the minimum permissible radius defined here, the control will display an error message. Input: **0.00000...999.99999**
B_MIN 	**Min. permissible width** If, after dressing, the actual width is below the minimum permissible width defined here, the control will display an error message. Input: **0.00000...999.99999**
V_MAX 	**Maximum permissible cutting speed** Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. Input: **0.000...999.999**
V	**Current cutting speed** Currently no function Input: **0.000...999.999**

Parameter	Meaning
W	**Tilt angle** Currently no function Input: **-90.00000...90.0000**
W_TYPE	**Tilted toward inner or outer edge** Currently no function Input: **-1**, **0**, **+1**
KIND	**Type of machining (internal/external grinding)** Currently no function Input: **0**, **1**
HW	**Wheel has a relief cut** Currently no function Input: **0**, **1**
HWA 	**Angle for relief cut on the outer edge** Input: **0.00000...45.00000**
HWI 	**Angle for relief cut on the inner edge** Input: **0.00000...45.00000**
INIT_D_OK	**Initial dressing performed** Initial dressing is the first dressing operation performed on the grinding wheel. If the following requirements are fulfilled, the control will set the parameter **INIT_D_OK** to **1**: ■ Grinding tool is defined ■ Initial dressing performed If the parameter **INIT_D_OK** is set to **1**, the control will disable the parameters for defining the grinding tool. If you set the parameter **INIT_D_OK** to **0**, the control will re-enable the editing of the parameters. In this case, the control will have to perform initial dressing of the tool again. **Further information:** User's Manual for Machining Cycles Input: **0**, **1**
INIT_D_PNR	**Dresser location for initial dressing** Currently no function Input: **0...9999**
INIT_D_DNR	**Dresser number for initial dressing** Currently no function Input: **0...32767**
MESS_OK	**Measure the grinding wheel** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Input: **0**, **1**
STATE	**Setup status** Currently no function Input: **%0000000000000000...%1111111111111111**

Parameter	Meaning
A_NR_D	**Dresser number (diameter dressing)** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Tool number of the dresser being used Corresponds to the **T_DRESS** parameter in the tool management Input: **0...32767**
A_NR_A	**Dresser number (outer edge dressing)** Currently no function Input: **0...32767**
A_NR_I	**Dresser number (inner edge dressing)** Currently no function Input: **0...32767**
DRESS_N_D	**Dressing counter for diameter (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the diameter. Input: **0...999**
DRESS_N_A	**Dressing counter for outer edge (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the outer edge. Input: **0...999**
DRESS_N_I	**Dressing counter for inner edge (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the inner edge. Input: **0...999**
DRESS_N_D_ACT	**Current dressing counter of the diameter** Current number of dressing cycles that have been skipped since the last dressing of the diameter. Input: **0...999**
DRESS_N_A_ACT	**Current dressing counter of the outer edge** Current number of dressing cycles that have been skipped since the last dressing of the outer edge. Input: **0...999**
DRESS_N_I_ACT	**Current dressing counter of the inner edge** Current number of dressing cycles that have been skipped since the last dressing of the inner edge. Input: **0...999**
AD	**Retraction amount at the diameter** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
AA	**Retraction amount at the outer edge** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**

Parameter	Meaning
AI 	**Retraction amount at the inner edge** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
FORM	**Wheel shape** Selection by means of a selection window Input: **0.00...99.99**
A_PL	**Chamfer length at outside** Input: **0.00000...999.99999**
A_PW	**Chamfer angle at outside** Input: **0.00000...89.99999**
A_R1	**Corner radius at outside** Input: **0.00000...999.99999**
A_L	**Length of outside** Currently no function Input: **0.00000...999.99999**
A_HL	**Length of relief cut, wheel depth at outside** Input: **0.00000...999.99999**
A_HW	**Angle of relief cut at outside** Input: **0.00000...45.00000**
A_S	**Side depth at outside** Depth of an already existing profile The control automatically corrects the value by the dressed value. Without function if **HWA** is defined Input: **0.00000...999.99999**
A_R2	**Angle of departure at outside** Input: **0.00000...999.99999**
A_G	**Reserve at outside** Currently no function Input: **0.00000...999.99999**
I_PL	**Chamfer length at inside** Input: **0.00000...999.99999**
I_PW	**Chamfer angle at inside** Input: **0.00000...89.99999**
I_R1	**Corner radius at inside** Input: **0.00000...999.99999**
I_L	**Length of inside** Currently no function Input: **0.00000...999.99999**
I_HL	**Length of relief cut, wheel depth at inside** Input: **0.00000...999.99999**
I_HW	**Angle of relief cut at inside** Input: **0.00000...45.00000**

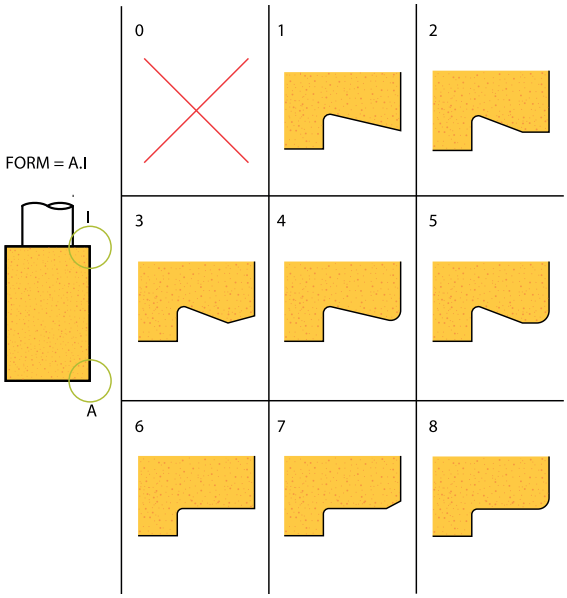
Parameter	Meaning
I_S	**Side depth at inside** Depth of an already existing profile The control automatically corrects the value by the dressed value. Without function if **HWI** is defined Input: **0.00000...999.99999**
I_R2	**Angle of departure at inside** Input: **0.00000...999.99999**
I_G	**Reserve at inside** Currently no function Input: **0.00000...999.99999**
COR_TYPE	**Selection of compensation method** You can choose between the following compensation methods: ■ **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** Compensation method with material removal at grinding tool **Further information:** Programming and Testing User's Manual ■ **Dressing tool with wear, COR_TYPE_DRESSTOOL** Compensation method with material removal at dressing tool Selection by means of a selection window Input: **0**, **1**
COR_ANG	**Inclination angle of dressing tool** Input: **0.00000**...**360.00000**

Shape of the grinding wheel face

You define the shape of the grinding wheel with the parameters of the geometric basic data. For the following grinding tool types you can additionally define the shape of the grinding wheel face for the front and shaft sides:

- **Cylindrical grinding pin**
- **Straight wheel**

The **Form** workspace assists you during program entry.



Selection possibilities for the grinding wheel face

The control offers a selection window where you select the combinations of grinding wheel faces. The first number defines the front face **A** and the second number defines the shaft face **I**. In each case you can choose the grinding wheel faces from **1** to **8**.

All selection possibilities except for **1** and **6** are complex shapes. If you choose a complex shape for one face of the grinding tool, then for the other face you can choose only **1** or **6**. In a selection window, the control displays first the possible combinations of complex shapes for the front face and then for the side face. Once you have chosen the shapes of the grinding wheel faces, the control displays only the parameters still needed.

- If you select a new grinding wheel face shape, the control removes all parameters not needed for that shape.
- Refer to your machine manual.
  The machine manufacturer can change the help graphics.

Defining the shape of the grinding wheel face (#156 / #4-04-1)

To define the front face **FORM 4** and shaft face **FORM 6**:

- Open the **Tables** operating mode
- Open **Tool management**
- Select or create the desired grinding tool
- Open the **Form** workspace
- Open the selection window in the **Shape of grinding wheel face** area
- Select **FORM 4.6**
- > The control displays the required parameters and the help graphics in the **Front face of grinding wheel** and **Shaft face of grinding wheel** areas.
- Define the required parameters for the grinding wheel shape in the form

Further information: "Setting up a tool", Page 107

Notes

- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with grinding tools.
- When you are dressing a tool, the control hides the tool carrier in the **Simulation** workspace.

  Further information: Programming and Testing User's Manual
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    Refer to your machine manual.

    If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.
- 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.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 493
- 3D radius compensation is not possible for grinding tools.

  Further information: Programming and Testing User's Manual
- The parameters required for the length depend on the type of grinding tool. The control filters the parameters in the **Form** workspace of the tool management depending on the selected tool type.

  There can be a radius **RV** at the edge of **L-OVR**. If you measure **L-OVR**, then do not take the radius **RV** into account. You measure **L-OVR** at the intersection of the adjoining teeth.

  Further information: Programming and Testing User's Manual
- If you have selected a grinding wheel edge shape, you can dress the radii **RV** and **RV1** with Cycle **1012 DRESSING D AND A/I**. To do so, set the parameters to these values:
  - **A_R1** = **RV**
  - **I_R1** = **RV1**

  The dressing cycle takes only the **A_R1** and **I_R1** parameters into account.

  Further information: User's Manual for Machining Cycles

19.5.5 Dressing tool table tooldress.drs (#156 / #4-04-1)

Application

The dressing tool table **tooldress.drs** contains the parameters specific to dressing tools.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool parameters
  Further information: "Tool parameters", Page 185
- Initial dressing
  Further information: User's Manual for Machining Cycles
- Grinding operations on milling machines
  Further information: Programming and Testing User's Manual
- Tool table for grinding tools
  Further information: "Grinding tool table toolgrind.grd (#156 / #4-04-1)", Page 473
- General parameters, regardless of the technology
  Further information: "Tool table tool.t", Page 457

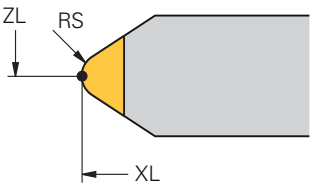
Requirements

- Grinding (#156 / #4-04-1) software option
- Dressing tool is defined in the **TYP** column of tool management
  Further information: "Tool types", Page 194

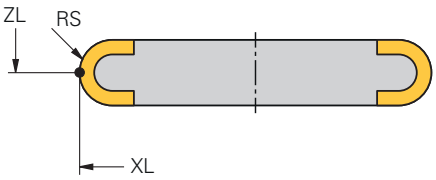
Description of function

The file name of the dressing tool table is **tooldress.drs** and this table must be stored in the folder **TNC:\table**.

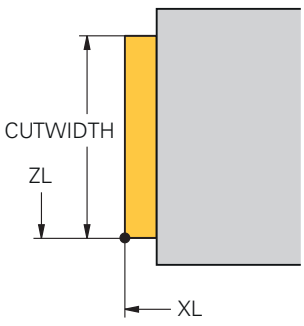
You define the parameters depending on the following dressing tool types:



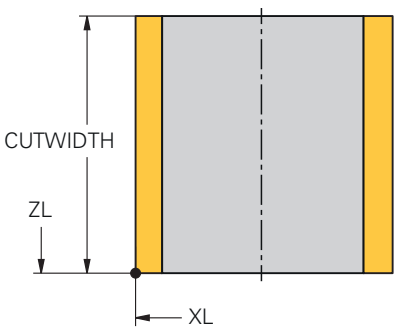
Stationary dresser with radius



Rotating dresser with radius



Stationary dresser (flat)



Rotating dresser (flat)

Parameters of the dressing tool table tooldress.drs

The **tooldress.drs** dressing tool table provides the following parameters:

Parameter	Meaning
T	Row number in the dressing tool table The tool number allows you to identify each tool unambiguously (e.g., for calling a tool). **Further information:** Programming and Testing User's Manual You can define an index after the period. **Further information:** "Indexed tool", Page 188 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Name of dressing tool** The tool name identifies a tool, for example when calling it. **Further information:** Programming and Testing User's Manual You can define an index after a period (i.e., name.index). **Further information:** "Indexed tool", Page 188 Input: **Text width 32**
ZL 	**Tool length 1** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 179 Input: **-99999.9999...+99999.9999**
DZL 	**Tool length oversize 1** Delta value of tool length 1 for tool compensation Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Tool length oversize 2** Delta value of tool length 2 for tool compensation Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
DYL 	**Tool length oversize 3** Delta value of tool length 3 for tool compensation Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Tool tip radius** Input: **0.0000...99999.9999**

Parameter	Meaning
DRS 	**Cutter radius oversize** Delta value of the cutter radius for tool compensation Is added to the parameter **RS** Input: **-999.9999...+999.9999**
TO 	**Tool orientation** The control uses the tool orientation to determine the position of the tool's cutting edge. Input: **1...9**
CUTWIDTH	**Width of tool (plate, roll)** Tool width of the tool types **dressing plate** and **dressing roll** Input: **0.0000...99999.9999**
TYPE 	**Type of dressing tool** Depending on the selected dressing tool type, the control displays the suitable parameters in the **Form** workspace of the tool management. **Further information:** "Dressing tool types (#156 / #4-04-1)", Page 196 **Further information:** "Tool management ", Page 197 Selection by means of a selection window Input: **DRESS_FIX_RADIUS**, **HORNED**, **DRESS_ROT_RADIUS**, **DRESS_FIX_FLAT** and **DRESS_ROT_FLAT**
N-DRESS	**Speed of the tool (dressing spindle)** Shaft speed of a dressing spindle or dressing roll Input: **0.0000...99999.9999**

Notes

- 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.
- When you are dressing a tool, the control hides the tool carrier in the **Simulation** workspace.

  Further information: Programming and Testing User's Manual
- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with dressing tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    Refer to your machine manual.

    If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.
- If you want to archive tool tables, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 493

19.5.6 Touch probe table tchprobe.tp

Application

In the touch probe table **tchprobe.tp** you define the parameters of the touch probe for the touching process, such as the probing feed rate. If you use several touch probes, you can save separate parameters for each touch probe.

Related topics

- Editing parameters in tool management
  Further information: "Tool management ", Page 197
- Tool parameters
  Further information: "Tool parameters", Page 185
- Touch probe functions
  Further information: "Touch probe functions in the Manual operating mode", Page 373
- Calibrating touch probe cycles for the workpiece touch probe
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Calibrating touch probe cycles for the tool touch probe
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Automatic touch probe cycles for the workpiece
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Automatic touch probe cycles for the tool
  Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Automatic touch probe cycles for measuring the kinematics
  Further information: Measuring Cycles for Workpieces and Tools User's Manual

Description of function

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

Parameters of the touch probe table tchprobe.tp

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	**Sequential number of touch probe** You enter this number in the **TP_NO** parameter in the tool management. The control then associates the data from the touch-probe table with the tool management. Input: **1...99**
TYPE 	**Selection of the touch probe?** The following values are available for the TS 642 touch probe: ■ **TS642-3**: The touch probe is activated by a conical switch. This mode is not supported. ■ **TS642-6**: The touch probe is activated by an infrared signal. Select this mode. Input: **TS120**, **TS220**, **TS249**, **TS260**, **TS440**, **TS444**, **TS460**, **TS630**, **TS632**, **TS640**, **TS642-3**, **TS642-6**, **TS649**, **TS740**, **TS 760**, **KT130**, **OEM**
STYLUS 	**Shape of the stylus** ■ **SIMPLE**: Straight stylus ■ **L-TYPE**: L-shaped stylus If you do not define the parameter, the control uses **SIMPLE**
CAL_OF1 	**TS center misalignmt. ref. axis? [mm]** Depending on the selection of **STYLUS** parameter, this parameter has the following function: ■ **SIMPLE**: Offset of the touch probe axis to the spindle axis in the main axis ■ **L-TYPE**: Length of extension on an L-shaped stylus Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle. Input: **-99999.9999...+99999.9999**
CAL_OF2 	**TS center misalignmt. aux. axis? [mm]** Offset of the touch probe axis to the spindle axis in the secondary axis Required when **ON** is selected in parameter **TRACK** The control describes this value in connection with the calibration cycle. Input: **-99999.9999...+99999.9999**
CAL_ANG 	**Spindle angle for calibration?** Depending on the selection of **STYLUS** parameter, this parameter has the following function: ■ **SIMPLE**: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible). ■ **L-TYPE**: The control orients the extension using the spindle angle. Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Required when **ON** is selected in parameter **TRACK** Input: **0.0000...359.9999**

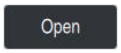
Parameter	Meaning
F 	**Probing feed rate? [mm/min]** In the machine parameter **maxTouchFeed** (no. 122602), the machine manufacturer defines the maximum probing feed rate. If **F** is greater than the maximum probing feed rate, then the maximum probing feed rate will be used. Input: **0...9999**
FMAX 	**Rapid traverse in probing cycle? [mm/min]** Feed rate at which the control pre-positions the touch probe and positions it between the measuring points Input: **+10...+99999**
DIST 	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will display an error message. Input: **0.00100...99999.99999**
SET_UP 	**Set-up clearance? [mm]** Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value. Input: **0.00100...99999.99999**
F_PREPOS 	**Pre-position at rapid? ENT/NOENT** Speed for pre-positioning: ■ **FMAX_PROBE**: Pre-position at the speed from **FMAX** ■ **FMAX_MACHINE**: Pre-position at machine rapid traverse Input: **FMAX_PROBE**, **FMAX_MACHINE**
TRACK 	**Probe oriented? Yes=ENT/No=NOENT** Orienting the infrared touch probe in each probing process: ■ **ON**: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ **OFF**: The control will not orient the touch probe. When selecting **L-TYPE** in the **STYLUS** parameter, **ON** must be selected. If you change the **TRACK** parameter, you must recalibrate the touch probe. Input: **ON**, **OFF**
SERIAL 	**Serial number?** The control automatically edits this parameter of touch probes with an EnDat interface. Input: **Text width 15**
REACTION 	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT** As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: ■ **NCSTOP**: Interrupt NC program ■ **EMERGSTOP**: Emergency stop, quick braking of the axes Input: **NCSTOP**, **EMERGSTOP**

Editing the touch probe table

To edit the touch probe table:

- Select the **Tables** operating mode

- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select the **tchprobe.tp** file in the **Open File** workspace



- Select **Open**
- > The control opens the **Touch probes** application.

- Activate **Edit**
- Select the desired value
- Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

19.5.7 Creating a tool table in inches

To create a tool table in inches:

- Select the **Manual** operating mode

- Select **T**
- Select the tool **T0**

- Press the **NC Start** key
> The control removes the current tool and does not insert a new tool.

- Restart the control
- Do not acknowledge **Power interrupted**

- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the original file (e.g., **tool.t** as **tool_mm.t**)

- Select the **Tables** operating mode

- Select **Create new table**
> The control opens the **Create new table** window.
- Select a folder with the corresponding table type (e.g., **t**)
- Select INCH as the unit of measure if necessary

- Select the desired prototype
- Select a path
> The control opens the **Save as** window.
- Select the **table** folder
- Enter a name (e.g., **tool**)

- Select **Create** twice
> The control opens the **Tool table** tab in the **Tables** operating mode.

- Restart the control

- Acknowledge **Power interrupted** with the **CE** key

- Select the **Tool table** tab in the **Tables** operating mode
> The control uses the newly created table as a tool table.

To use the **Tool management** application you have to create all existing tool tables in inches.

19.6 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine.
The control needs the pocket table in order to change the tool.

Related topics

- Tool call
  Further information: Programming and Testing User's Manual
- Tool table
  Further information: "Tool table tool.t", Page 457

Requirement

- The tool is defined in the tool management.
  Further information: "Tool management ", Page 197

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	**Pocket number?** Pocket number of the tool in the tool magazine Input: **0.0...99.9999**
T	**Tool number?** Row number of the tool from the tool table With the machine parameter **deleteLoadedTool** (no. 125301) you define whether you are allowed to edit the **T** column. The machine manufacturer enables this parameter. **Further information:** "Tool table tool.t", Page 457 Input: **1...99999**
TNAME	**Tool name?** Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. **Further information:** "Tool table tool.t", Page 457 Input: **Text width 32**
RSV	**Reserve pocket?** When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: ■ No value entered: Pocket is not reserved ■ **R**: Pocket is reserved Input: No value, **R**

Parameter	Meaning
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ **S**: Special tool Input: No value, **S**
F	**Fixed pocket?** Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: ■ No value entered: No fixed pocket ■ **F**: Fixed pocket Input: No value, **F**
L	**Locked pocket?** To lock a pocket for tools (e.g., the pockets next to special tools): ■ No value entered: Do not lock ■ **L**: Lock Input: No value, **L**
DOC	**Pocket comment?** The control automatically loads the tool comment from the tool table. **Further information:** "Tool table tool.t", Page 457 Input: **Text width 32**
PLC	**PLC status?** Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **%00000000...%11111111**
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **-99999.9999...+99999.9999**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **0...99**
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**

Parameter	Meaning
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LAST_USE	**LAST_USE** The control automatically loads the date and time of the last tool call from the tool table. **Further information:** "Tool table tool.t", Page 457 Refer to your machine manual. Entry: **Text width 20**
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**

19.7 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
  Further information: "Tool usage test", Page 208
- Working with a pallet table
  Further information: Programming and Testing User's Manual
- Tool data from the tool table
  Further information: "Tool table tool.t", Page 457

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
  In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.
  Further information: "Creating the tool usage file", Page 208
- The **Generate tool-usage file** setting is set to **Once** or **Always**
  Further information: "Channel Settings", Page 562

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file Input: **0...99999**
TOKEN	In the **TOKEN** column, the control uses one word to show which information is contained in the respective row: ■ **TOOL**: Data per tool call; listed in chronological order ■ **TTOTAL**: All data of a tool; listed in alphabetical order ■ **STOTAL**: Called NC programs; listed in chronological order ■ **TIMETOTAL**: Total tool usage time of an NC program ■ **TOOLFILE**: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table **tool.t** Input: **Text width 17**
TNR	Tool number If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767**
IDX	Tool index Input: **0...9**
NAME	Tool name Input: **Text width 32**
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Input: **0...9999999**
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece Input: **0...9999999**
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table Input: **-999999.9999...999999.9999**
BLOCK	NC block number of the tool call Input: **0...999999999**
PATH	Path of the NC program, the pallet table, or the tool table Input: **Text width 300**
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767.9**

Parameter	Meaning
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**. Input: **0...32767**
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**. Input: **-1...32767**
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed Input: **0**, **1**
LINENR	Row number of the pallet table in which the NC program is defined Input: **-1...99999**

Note

The control saves the tool usage file as a dependent file (***.dep**).

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

19.8 T usage order (#93 / #2-03-1)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 208

  Further information: "Tool usage file", Page 497

Description of function

When you select an NC program in the **Program Run** operating mode, the control will automatically create the **T usage order** table. The control displays the table in the **T usage order** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot edit the table.

The **T usage order** table provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 188 May differ from the programmed tool (e.g., when a replacement tool is used)
NAME	Name of the tool used, including an index as needed **Further information:** "Indexed tool", Page 188 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 494 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool management ", Page 197
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 188
USAGE	Total tool usage time from the **WTIME** column of the **tool usage file** (in seconds) Total time between the tool changes, during which the tool is cutting a workpiece **Further information:** "Tool usage file", Page 497
TOOL TIME	Estimated time of tool change

Parameter	Meaning
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 497
MIN OVRD	Minimum value of the feed-rate potentiometer during program run (in percent)
MAX OVRD	Maximum value of the feed-rate potentiometer during program run (in percent)
NC PGM	Path of the NC program in which the tool is programmed
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 494

19.9 Tooling list (#93 / #2-03-1)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 208

  Further information: "Tool usage file", Page 497

Description of function

When you select an NC program in the **Program Run** operating mode, the control will automatically create the **Tooling list** table. The control displays the table in the **Tooling list** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in numerical order. You cannot edit the table.

The **Tooling list** table provides the following parameters:

Parameter	Meaning
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 188 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 494 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 201
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 188
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 497
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 494

19.10 Preset table *.pr

Application

The **preset.pr** preset table allows you to manage presets, such as the position and misalignment of a workpiece in the machine. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 174

Related topics

- Setting and activating presets

  Further information: "Preset management", Page 230

Description of function

By default, the preset table has the name **preset.pr**, and is saved in the **TNC:\table** directory. The preset table is open in the **Tables** operating mode by default.

Refer to your machine manual.

The machine manufacturer can define a different path for the preset table.

In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
	Active row
	Write-protected row

When you define a preset, the control opens a window with the following input options:

Button	Function
Set a preset	The control interprets the entered value as desired display value for the actual position. The control calculates the required table value from this. The entered value is active in the basic coordinate system **B-CS**. **Further information:** "Basic coordinate system B-CS", Page 219 When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value. The entered value is active incrementally in the basic coordinate system **B-CS**.
Edit	The control accepts the entered value unchanged as table value. The entered value refers to the coordinate origin of the basic coordinate system **B-CS**.

Parameters of the preset table

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row Input: **0...99999999**
DOC	Comment Entry: **Text width 16**
X	X coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Input: **-99999.99999...+99999.99999**
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Input: **-99999.99999...+99999.99999**
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Input: **-99999.99999...+99999.99999**
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 232 Input: **-99999.9999999...+99999.9999999**
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 232 Input: **-99999.9999999...+99999.9999999**
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 219 Has the effect of a basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 232 Input: **-99999.9999999...+99999.9999999**
X_OFFS	Position of the X axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**
Y_OFFS	Position of the Y axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**

Parameter	Meaning
Z_OFFS	Position of the Z axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**
A_OFFS	Axis angle of the A axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.9999999...+99999.9999999**
B_OFFS	Axis angle of the B axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.9999999...+99999.9999999**
C_OFFS	Axis angle of the C axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.9999999...+99999.9999999**
U_OFFS	Position of the U axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 216 Input: **-99999.99999...+99999.99999**
ACTNO	Active workpiece preset The control automatically enters **1** in the active row. Input: **0**, **1**
LOCKED	Write-protection of the table row Entry: **Text width 16**

Refer to your machine manual.
In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Basic transformation and offset

The control interprets the basic transformations **SPA**, **SPB** and **SPC** as basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. During program execution, the control moves the linear axes in accordance with the basic rotation without any change in the workpiece position.

Further information: "Basic rotation and 3D basic rotation", Page 232

The control interprets all offsets for each respective axis as a shift in the machine coordinate system **M-CS**. The effect that offsets have is contingent on the kinematics.

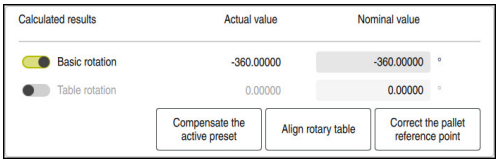
Further information: "Machine coordinate system M-CS", Page 216

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** probing function to determine the misalignment of a workpiece. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating the basic rotation of a workpiece", Page 387



Results of the **Rotation (ROT)** probing function

If you activate the **Basic rotation** toggle switch, the control interprets the misalignment as a basic transformation. When using the **Compensate the active preset** button, the control saves the result in the columns **SPA**, **SPB** and **SPC** of the preset table. The **Align rotary table** button has no function in this case.

If you activate the **Table rotation** toggle switch, the control interprets the misalignment as an offset. When using the **Compensate the active preset** button, the control saves the result in the columns **A_OFFS**, **B_OFFS** and **C_OFFS** of the preset table. To move the rotary axes to the position of the offset, use the **Align rotary table** button.

Write-protection for table rows

The **Lock record** button allows protecting any rows of the preset table against overwriting. The control enters the value **L** in the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 508

Alternatively, the row can be protected with a password. The control enters the value **###** into the **LOCKED** column.

Further information: "Protecting table rows with a password", Page 508

The control displays an icon ahead of write-protected rows.

If the control displays the value **OEM** in the **LOCKED** column, this column has been locked by the machine manufacturer.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

19.10.1 Activating write protection

Protecting table rows without a password

To protect a table row without a password:

- Activate the **Edit** toggle switch
- Select the desired row

- Activate the **Lock record** toggle switch
- > The control enters the value **L** in the **LOCKED** column.

- > The control activates write-protection and displays an icon ahead of the row.

Protecting table rows with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

To protect a table row with a password:

- Activate the **Edit** toggle switch
- Double-tap or double-click the **LOCKED** column of the desired row
- Enter the password
- Confirm your input
- > The control enters the value **###** in the **LOCKED** column.

- > The control activates write-protection and displays an icon ahead of the row.

19.10.2 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

- Activate the **Edit** toggle switch

- Deactivate the **Lock record** toggle switch
- > The control removes the value **L** from the **LOCKED** column.
- > The control deactivates the write protection and removes the icon ahead of the row.

Unlocking table rows that are protected with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

To unlock a table row that is protected with a password:

- Activate the **Edit** toggle switch
- Double-tap or double-click the **LOCKED** column of the desired row
- Delete ***###***
- Enter the password
- Confirm your input
- > The control deactivates write-protection and removes the icon ahead of the row.

19.10.3 Creating a preset table in inches

If you define inches as the unit of measure in the **Machine Settings** menu item, the unit of measure of the preset table will not be adjusted automatically.

Further information: "The Machine Settings menu item", Page 562

To create a preset table in inches:

- Restart the control
- Do not acknowledge **Power interrupted**

- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the original file **preset.pr** (e.g., as **preset_mm.pr**)

- Select the **Tables** operating mode

- Select **Create new table**

> The control opens the **Create new table** window.

- Select the **pr** folder
- Select INCH as the unit of measure if necessary

- Select the desired prototype
- Select a path

> The control opens the **Save as** window.

- Select the **table** folder
- Enter the name **preset.pr**

- Select **Create** twice

> The control opens the **Presets** tab in **Tables** operating mode.

- Restart the control

- Acknowledge **Power interrupted** with the **CE** key

- Select the **Presets** tab in **Tables** operating mode

> The control uses the newly created table as a preset table.

> The control shows INCH as the unit of measure in the dialog bar of the workspaces.

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- Before activating a preset, check whether all columns contain values.
- For undefined columns, enter values (e.g., **0**)
- As an alternative, have the machine manufacturer define **0** as the default value for the columns

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- If you edit the value of the **DOC** column, then the preset must be reactivated. Only then does the control apply the new value.

  Further information: "Activating presets", Page 231
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

  Further information: Programming and Testing User's Manual
- If a manual probing process or an NC program is interrupted or stopped, you cannot edit the preset table. When you double-tap or double-click a table cell the control shows the **Editing not possible. Perform internal stop?** window. If you select **Yes**, the control may lose touch points or modally active program information.

Notes about machine parameters

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- If the unit of measure of the preset table does not match the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the dialog bar of the **Tables** operating mode.
- The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offsets in the following NC functions:
  - **FUNCTION PARAXCOMP**
  - **POLARKIN** (#8 / #1-01-1)
  - **FUNCTION TCPM** or **M128** (#9 / #4-01-1)
  - **FACING HEAD POS** (#50 / #4-03-1)

    Further information: Programming and Testing User's Manual

19.11 Tables for AFC (#45 / #2-31-1)

19.11.1 Basic AFC settings in AFC.tab

Application

In the **AFC.tab** table, you define the feed-rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Related topics

- Programming AFC

  Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

The data in this table are default values that, during a teach-in cut, are copied into an associated dependent file of the relevant NC program. The values are the basis for feedback control.

Further information: "Description of function", Page 515

If you define a tool-specific reference power in the **AFC-LOAD** column in the tool table, the control will create the associated dependent file for the respective NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:

Parameter	Meaning
NR	Row number in the table Input: **0...9999**
AFC	Name of the control setting Enter this name in the **AFC** tool management column. It specifies the assignment of the control parameters to the tool. Entry: **Text width 10**
FMIN	Feed rate at which the control will perform an overload response Enter the value in percent of the programmed feed rate Not necessary in turning mode (#50 / #4-03-1) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 284 Input: **0...999**
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate Not necessary in turning mode (#50 / #4-03-1) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 284 Input: **0...999**
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (#50 / #4-03-1) Input: **0...999**
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (#50 / #4-03-1) Input: **0...999**

Parameter	Meaning
OVLD	Desired reaction of the control to overload: ■ **M**: Execution of a macro defined by the machine manufacturer ■ **S**: Immediate NC stop ■ **F**: Execute NC stop when the tool is no longer in the material ■ **E**: Just display an error message on the screen ■ **L**: Disable active tool ■ **-**: No overload reaction If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum while feedback control is active, the control will conduct an overload reaction. In conjunction with the cut-related tool wear monitoring function, the control will evaluate only the options **M**, **E**, and **L**! For tool-load monitoring with the column **AFC_OVLD2**, this parameter has no function. Input: **M**, **S**, **F**, **E**, **L**, or **-**
POUT	Spindle power at which the control will detect that the tool exits the workpiece Enter the value in percent of the learned reference load Recommended input value: 8% In turning mode: Minimum load **Pmin** for tool monitoring (#50 / #4-03-1) Input: **0...100**
SENS	Sensitivity (aggressiveness) of feedback control 50 is for slow feedback control, 200 for a very aggressive feedback control. An aggressive feedback control responds quickly and significantly changes the values, but it tends to overshoot. In turning mode: Activate the monitoring of the minimum load **Pmin** (#50 / #4-03-1): ■ **1**: Evaluate **Pmin** ■ **0**: Do not evaluate **Pmin** Input: **0...999**
PLC	Value that the control will transfer to the PLC at the beginning of a machining step The machine manufacturer defines whether and which function will be performed by the control. Input: **0...999**

Notes

- If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a permanently defined, internal control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends using the AFC.TAB table in order to ensure safe and well-defined operation.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

  Further information: Programming and Testing User's Manual

19.11.2 AFC.DEP settings file for teach-in cuts

Application

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

Related topics

- AFC basic settings in the table **AFC.tab**
  Further information: "Basic AFC settings in AFC.tab", Page 512
- Setting up and using AFC
  Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a ***** in front of these control settings in the AFC column.

Further information: "Basic AFC settings in AFC.tab", Page 512

In addition to the contents from the **AFC.tab** table, the **AFC.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed (not editable) In conjunction with AFC (#45 / #2-31-1), the following characters are not permitted in the tool name: # $ & , .
IDX	Index of the tool with which the machining step was performed (not editable)
N	Difference for tool call: ■ **0**: Tool was called by its tool number ■ **1**: Tool was called by its tool name
PREF	Reference load of the spindle. The control measures the value in percent with respect to the rated spindle power
ST	Status of the machining step: ■ **L**: In the next program run, a teach-in cut is recorded for this machining step. The control overwrites any existing values in this row ■ **C**: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Notes

- Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

  The control does not remove the editing lock until one of the following functions has been executed:

  - **M2**
  - **M30**
  - **END PGM**
- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

  Further information: Programming and Testing User's Manual

19.11.3 Log file AFC2.DEP

Application

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. During feedback control, the control updates the data and performs various evaluations.

Related topics

- Setting up and using AFC

  Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 276

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

The **AFC2.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed
IDX	Index of the tool with which the machining step was performed
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power

Column	Function
OVLD	Overload reaction performed by the control: ■ **M**: A macro defined by the machine manufacturer has been run ■ **S:** Immediate NC stop was conducted ■ **F**: NC stop was performed once the tool was no longer in the material ■ **E**: An error message was displayed ■ **L**: The current tool was locked ■ **-**: There was no overload response
BLOCK	Block number at which the machining step begins

During feedback control, the control determines the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Note

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

19.11.4 Editing the tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of the active NC program.

To open a table for AFC:

- Select the **Program Run** operating mode

- Select **AFC settings**
- > The control displays a selection menu. The control displays all the tables available for this NC program.
- Select a file (e.g., **AFC.TAB**)
- > The control opens the file in the **Tables** operating mode.

20

Electronic handwheel

20.1 Fundamentals

Application

The electronic handwheel allows you to traverse the axes without needing to stand at the operating panel. You can also use the handwheel to perform control functions, such as setting up the machine or influencing program run.

Refer to your machine manual.
The machine manufacturer defines which functions are available on the handwheel and how the control evaluates these functions. This chapter describes the standard features of the handwheel.

Related topics

- Incremental jog positioning
  Further information: "Incremental jog positioning of axes", Page 165
- Handwheel superimpositioning with GPS (#44 / #1-06-1)
  Further information: "The Handwheel superimp. function", Page 296
- Handwheel superimpositioning with **M118**
  Further information: Programming and Testing User's Manual
- Virtual tool axis **VT** (#44 / #1-06-1)
  Further information: Programming and Testing User's Manual
- Touch probe functions in **Manual** operating mode
  Further information: "Touch probe functions in the Manual operating mode", Page 373

Overview

The control supports the following handwheels:

Handwheel	Meaning	Further information
HR 130, HR 180	Handwheel mounted on the operating panel	
HR 510, HR 510 FS	Handwheel without display	Page 522
HR 520, HR 520 FS HR 550 FS	Handwheel with display with wireless transmission	Page 525

Notes

⚠ DANGER

Caution: hazard to the user!

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

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

- The machine manufacturer defines which axes you can move with the handwheel. Your machine manufacturer can also place the virtual axis **VT** on an axis key.
- If the handwheel is active, the control shows an icon for the selected axis in the **Positions** workspace. The icon indicates whether you can move the axis with the handwheel.

  Further information: "The Positions workspace", Page 119

20.2 Handwheel without display

Application

This chapter contains supplementary information about the HR 510 and HR 510 FS handwheels without display.

Related topics

- Overview of the available handwheels
  Further information: "Overview", Page 520
- Handwheels with display
  Further information: "Handwheel with display", Page 525

Description of function

Operating elements

A handwheel without display features the following operating elements:

1 Axis keys
2 The **actual position capture** key
  Further information: "Creating an NC block with the current position", Page 524
3 Keys for the speed level
  Further information: "Speed levels", Page 523
4 Traverse direction keys
5 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** and **NC stop**
6 Handwheel permissive buttons
  Further information: "Activating and deactivating a handwheel", Page 523
7 Dial
8 The **Emergency stop** button

Speed levels

The handwheel offers three keys with pre-defined speed levels that you can choose from.

The speed level influences the following values:

- Distance that the control moves an axis by when you rotate the wheel by one detent stop
- Feed rate at which the control moves the axis when you press an axis-direction key

You use a key to specify both the feed rate as well as the speed level for the distance moved. However, the control uses different, independent values, depending on whether you rotate the wheel or press a key.

Key	Meaning
	Low speed level Example: Distance of 0.001° or 0.001 mm/inch
	Medium speed level Example: Distance of 0.01° or 0.01 mm/inch
	High speed level Example: Distance of 0.1° or 0.1 mm/inch

Refer to your machine manual.

The machine manufacturer defines the values of the speed levels for each axis.

20.2.1 Activating and deactivating a handwheel

To activate a handwheel without display:

- ▶ Press the **Handwheel** key on the control
- > The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
- > The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.

To deactivate a handwheel without display:

- ▶ Press the **Handwheel** key on the control

20.2.2 Creating an NC block with the current position

Refer to your machine manual.
The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:

- Select the **Manual** operating mode
- Select the **MDI** application
- If necessary, select the NC block after which you want to insert the NC block
- Activate the handwheel
- Press the **actual position capture** key
  > The control inserts a straight line **L** with the actual positions of all defined axes.

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**.

20.2.3 Incremental jog positioning using handwheels without display

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key. In order to perform incremental jog positioning using a handwheel without display, you must first define the jog increment on the control.

Incremental jog positioning works only for traverse movements using the axis-direction keys.

To perform incremental jog positioning using a handwheel without display:

- Select the **Manual** operating mode
- Select the **Manual operation** application
- Select **Jog increment**
  > The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
- Enter the jog increment for linear axes and rotary axes
- Press the **Handwheel** key on the control
  > The control activates the handwheel.
- Press an axis key
- Press a traverse direction key
  > The control moves the axis by the defined jog increment.

20.3 Handwheel with display

Application

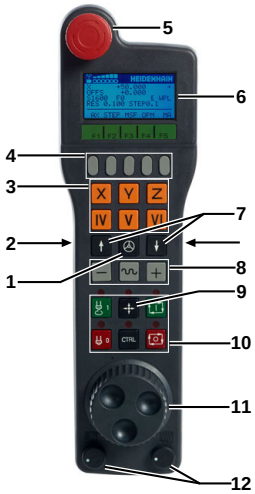
This chapter contains information specific to the HR 520, HR 520 FS and HR 550 FS handwheels with display. Handwheels with display can perform more functions than handwheels without display.

Related topics

- Overview of handwheels
  Further information: "Overview", Page 520
- Handwheels without display
  Further information: "Handwheel without display", Page 522

Description of function

Operating elements

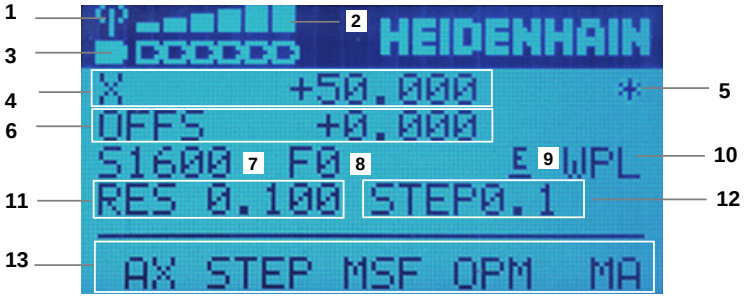


A handwheel with display provides the following operating elements:

1 Handwheel activation key
Further information: "Activating and deactivating a handwheel with display", Page 530
2 Handwheel permissive buttons on the sides
3 Axis keys
4 Soft-key selection keys
Further information: "Handwheel soft keys", Page 527
5 The **Emergency stop** button
6 Display
Further information: "Contents of display", Page 526
7 Speed level
Further information: "Speed levels", Page 528
8 Keys for traverse direction and rapid traverse
9 The **actual position capture** key
Further information: "Creating an NC block with the current position", Page 524

10 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** and **NC stop**
11 Dial
12 Potentiometers for spindle speed and feed rate

Contents of display



The display of a handwheel consists of the following areas:

1 Handwheel is in the docking station or radio mode is active
Only with HR 550 FS wireless handwheel
2 Field strength, up to six bars
Only with HR 550 FS wireless handwheel
3 Charge status of the rechargeable battery, up to six battery symbols
Only with HR 550 FS wireless handwheel
4 Selected axis and current position
5 Control-in-operation
Program run has been started or axis is in motion
6 Handwheel superimpositioning from **M118** or the Global Program Settings GPS (#44 / #1-06-1)
Further information: "The Handwheel superimp. function", Page 296
Further information: Programming and Testing User's Manual
7 Current speed of the active spindle
8 Current feed rate of the selected axis
During program run: current contouring feed rate
9 Pending error message
10 Active setting in the **3-D rotation** window:
- **VT**: **Tool axis** function
- **WP**: **Basic rotation** function
- **WPL**: **3D ROT** function

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 239
11 Traverse per wheel detent stop
Further information: "Speed levels", Page 528
12 Incremental jog active or inactive, and jog increment
Further information: "Incremental jog positioning", Page 532
13 Handwheel soft keys
Further information: "Handwheel soft keys", Page 527

Handwheel soft keys

You can use the handwheel soft keys to select the following functions:

Soft key	Key	Meaning
AX	**F1**	Select machine axis
STEP	**F2**	Active or deactivate incremental jog positioning, and select the jog increment **Further information:** "Incremental jog positioning", Page 532
MSF	**F3**	Define cutting data, miscellaneous functions, and presets **Further information:** "Defining the spindle speed S", Page 531
OPM	**F4**	Select operating mode
MA	**F5**	Execute machine-specific functions (such as switching magazine pockets)
MOP	**F3**	Select manual options Only if program run was interrupted by an **NC stop**

Operating modes

Press **OPM** to choose between the following operating modes:

Soft key	Key	Meaning
MAN	**F1**	**Manual** operating mode
MDI	**F2**	**MDI** application in the **Manual** operating mode
RUN	**F3**	**Program Run** operating mode
SGL	**F4**	**Single Block** mode in the **Program Run** operating mode

Functions during program run

Press **MOP** to choose between the following functions:

Soft key	Key	Meaning
MAN	**F1**	Manual traverse
STOP	**F4**	Internal stop

Press **MAN** at this level to choose between the following functions:

Soft key	Key	Meaning
REPO	**F1**	Return to the contour
3D	**F2**	Settings of the Tilt working plane function

Speed levels

You use the up and down arrow keys to choose the speed level.

The handwheel shows the value of the selected level in the display after **RES**. If you rotate the wheel by one detent stop, the control moves the axis by the value in the unit of measure for that axis.

Key	Meaning
↑	Increase speed level
↓	Decrease speed level

On handwheels with display the speed level defines only the distance per detent stop; it does not define the feed rate for traverse movements with axis-direction keys.

You use the **MSF** soft key to define the feed rate.

Further information: "Defining the handwheel feed rate F", Page 531

Special features of the HR 550 FS wireless handwheel

With the HR 550 FS wireless handwheel you can move farther away from the machine operating panel than with cable-connected handwheels. The HR 550 FS wireless handwheel thus provides an important benefit, in particular for large machines.

The HRA 551 FS handwheel holder and the HR 550 FS handwheel together form one functional unit.

HR 550 FS handwheel

HRA 551 FS handwheel holder

The HR 550 FS wireless handwheel features a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HR 550 FS can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you aren't using the HR 550 FS, always place it in the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.

Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.

You can configure wireless handwheels in the **TNCdiag** application.

Further information: "Setting up a wireless handwheel ", Page 534

If the control has triggered an emergency stop, you must reactivate the handwheel.

If you happen to get close to the limit of the transmission range, the HR 550 FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

20.3.1 Activating and deactivating a handwheel with display

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- Retract the tool before switching
- Before switching, check whether the settings of the feed-rate potentiometers match

To activate a handwheel with display:

- Press the **Handwheel** key on the handwheel
- > The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
- > The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.
- If necessary, adjust the setting of the feed-rate potentiometer

To deactivate a handwheel with display:

- Press the **Handwheel** key on the handwheel

The control also activates or deactivates the feed-rate potentiometer.

If the feed rate before switching is higher than the feed rate after switching, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching is less than the feed rate after switching, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because only then will the activated feed-rate potentiometer take effect.

20.3.2 Creating an NC block with the current position

Refer to your machine manual.

The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:

- Select the **Manual** operating mode
- Select the **MDI** application
- If necessary, select the NC block after which you want to insert the NC block
- Activate the handwheel

- Press the **actual position capture** key
- > The control inserts a straight line **L** with the actual positions of all defined axes.

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**.

20.3.3 Defining the spindle speed S

To define the spindle speed **S** of the active spindle when using a handwheel with display:

- Press the **MSF** soft key on the handwheel
- Press the **S** soft key on the handwheel
- Select the desired spindle speed by pressing the **F1** or **F2** keys
- > The handwheel shows the defined spindle speed in the display after **S**.
- Press the **NC Start** key
- > The control activates the defined spindle speed.

If you press and hold the **F1** or **F2** key, the handwheel counts the value up or down. The longer you press the key, the greater the counting step.

If you additionally press the **CTRL** key, the handwheel starts with a larger counting step.

20.3.4 Defining the handwheel feed rate F

To define the feed rate **F** when using a handwheel with display:

- Press the **MSF** soft key on the handwheel
- Press the **F** soft key on the handwheel
- Select the desired feed rate by pressing the **F1** or **F2** keys
- Press **OK**
- Confirm the new feed rate by pressing the **OK** soft key on the handwheel

X
- Press an axis key

+
- Press a traverse direction key
- > The control moves the axis at the defined feed rate.

If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a value divisible by 10.

By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

20.3.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key.

- Incremental jog positioning works only for traverse movements using the axis-direction keys.
- The control compares the settings for incremental jog positioning defined for the handwheel and the control.

To perform incremental jog positioning using a handwheel with display:

- ▶ Press the **STEP** soft key on the handwheel
- ▶ Press the **ON** soft key on the handwheel
- > The control activates incremental jog positioning.
- ▶ Select the desired jog increment by pressing the **F1** or **F2** keys
- > The handwheel shows the defined jog increment in the display after **STEP**.

The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

- ▶ Confirm the jog increment by pressing the **OK** soft key on the handwheel

- ▶ Press an axis key

- ▶ Press a traverse direction key
- > The control moves the axis by the defined jog increment.

If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a value divisible by 10.

By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes on wireless handwheels

⚠ DANGER

Caution: hazard to the user!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- Check the radio connection of the handwheel for possible overlapping with other wireless devices
- Switch off the handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test the next time it is restarted (only for handwheels 598515-03, 606622-03 and holder 731928-02)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

NOTICE

Caution: Danger to the tool and workpiece!

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

- Place the handwheel in the handwheel holder when it is not in use
- Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
- Test the handwheel before machining

- The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

20.4 Setting up a wireless handwheel

Application

You can configure the HR 550 FS wireless handwheel in the **Setup for wireless handwheel** application.

Related topics

- Electronic handwheel
  Further information: "Electronic handwheel", Page 519
- HR 550 FS wireless handwheel
  Further information: "Special features of the HR 550 FS wireless handwheel", Page 529
- TNCdiag
  Further information: "TNCdiag", Page 623

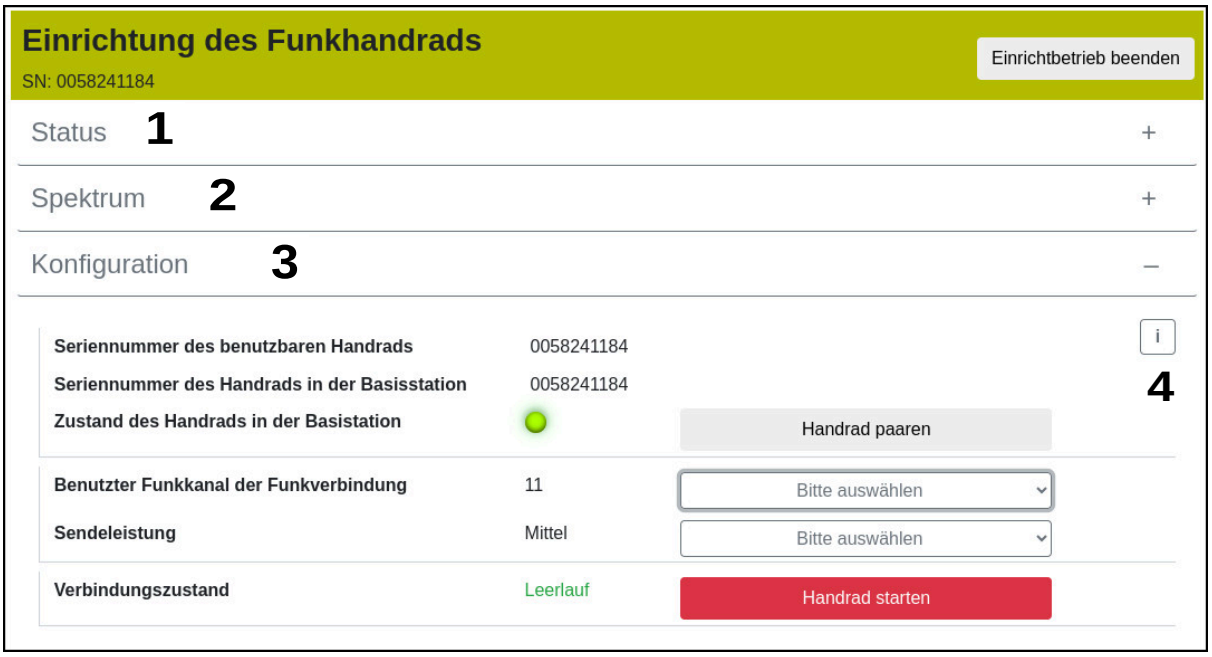
Requirements

- Machine with a handwheel holder
  The machine manufacturer installs the handwheel holder on the machine.

Description of function

To navigate to this function:

Home ▶ **Settings** ▶ **Machine Settings** ▶ **Set Up Wireless Handwheel**



TNCdiag within the **Setup for wireless handwheel** application

TNCdiag shows the following areas:

1 **Status**

Information about the transmission quality

If the reception quality of the wireless connection is poor, the control triggers an emergency stop. Safe stopping of the axes cannot be ensured when the reception quality is poor.

2 **Spectrum**

Frequency of the individual radio channels

The radio channel with the shortest column has the least amount of radio traffic. The radio channel recommend for the wireless handwheel is marked with a green circle.

3 **Configuration**

- **Pair handwheel**

  Assign the wireless handwheel to the handwheel holder
- **Channel used for radio connection**

  In the selection menu, choose **Best channel** to select the recommend radio channel for the wireless handwheel
- **Transmitter power**

  Select Transmitter power in the selection menu. The lower the transmission power, the smaller the range of the wireless handwheel.
- **Connection status**

  Once **TNCdiag** continuously shows **Active** as the connection status, configuration is complete.

4 **Information**

Each area shows the **Information** icon. When you select the icon, **TNCdiag** displays the description of the settings.

Setting up a new wireless handwheel

To set up a new wireless handwheel:

- Place the handwheel in the handwheel holder

- Select the **Home** operating mode

- Select the **Settings** application

- Select **Machine Settings**

- Double-tap or double-click **Set Up Wireless Handwheel**
- > The control opens the **Setup for wireless handwheel** application within **TNCdiag**.
- Select **Pair handwheel**
- > **TNCdiag** briefly shows **Active** under **Connection status**.
- In the **Channel used for radio connection** selection menu, choose **Best channel** for the channel
- In the **Transmitter power** selection menu, choose the transmitter power (e.g. **Medium**)
- Select **Start handwheel**
- > **TNCdiag** activates the handwheel.
- > **TNCdiag** dims the **Channel used for radio connection** and **Transmitter power** selection menus.

If an error occurs during configuration, **TNCdiag** colors the **Configuration** area red.

In order to read the error details, switch to an operating mode, such as the **Home** operating mode.

21

Override controller

Application

The override controller is an operating element with additional functions compared to a usual override potentiometer.

In conjunction with the override controller, the control gives you the following possibilities:

- Use the dial to manipulate the feed rate and/or rapid traverse
- Start NC programs with the integrated **NC Start** button
- Receive tactile responses through vibrations
- Use breakpoints to define conditional stops
- Resume the NC program by increasing the override

Requirements

- Override controller OC 310

  The availability of the override controller depends on the machine.

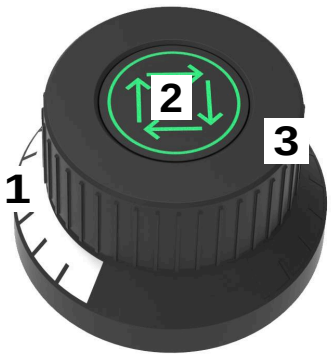
  Refer to your machine manual.
- Control is fully booted

  The control only detects the override controller once the machine control voltage has been acknowledged.
- Tool inspection has been performed

  Further information: "The Tool check column in the Program workspace", Page 209

Description of function

Elements of the override controller



The override controller consists of the following elements:

1 Override scale

  The override scale is illuminated in color up to the current override value.

  Further information: "Visual feedback from the override controller", Page 539

2 The **NC Start** button

The **NC Start** button starts the NC program.

Depending on the setting in the **Program run options** window, the NC program can be continued with the **NC Start** button.

3 Dial

Use the dial to change the override for the feed rate and/or rapid traverse.

Depending on the setting in the **Program run options** window, the NC program can be continued with the Override.

Visual feedback from the override controller

The override controller uses the following visual feedback:

Status	Override scale
Override Controller not active (e.g., because of an emergency stop)	Not illuminated
Override value of 0%	Not illuminated
Override value between 0% and 99.5%	White
Override value of 100%	Green
Override value greater than 100.5%	Blue

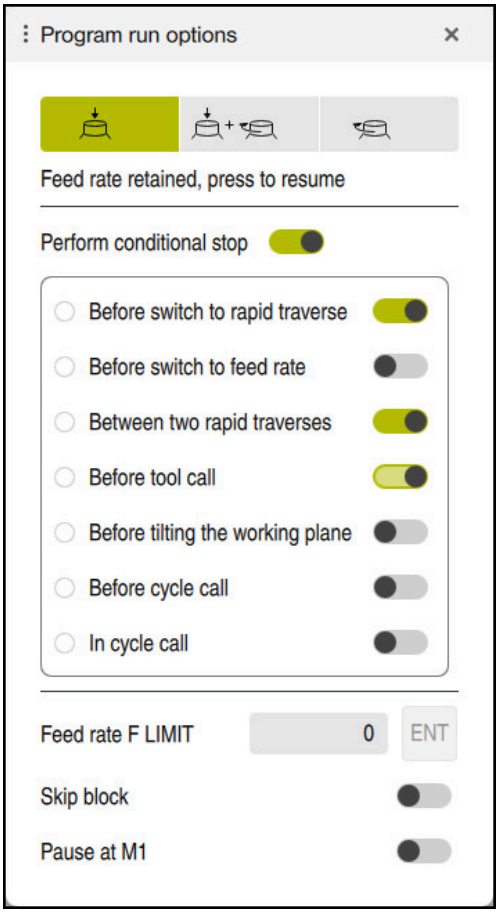
The **NC Start** button lights up green. The actual color may differ, depending on the machine.

Tactile feedback from the override controller

The override controller uses the following tactile feedback:

Status	Acknowledgment
Minimum or maximum override value	The override controller vibrates as soon as the minimum or maximum override value is reached.
Override value of 100%	The override controller vibrates as soon as the override value is at 100%.
Stop at the breakpoint	The override controller vibrates as soon as the control stops at a breakpoint.

The Program run options window



The **Program run options** window

You can open the **Program run options** window as follows:

- In the **Program Run** operating mode with the **Program run options** button

  Further information: "Icons and buttons", Page 414
- In the **Simulation** workspace with the **Program run options** toggle switch in the **Visualization options** column

  Further information: Programming and Testing User's Manual

The following settings of the **Program run options** window are relevant for the override controller:

Icon or button	Meaning
	Feed rate retained, press to resume When this button is active, the control does not change the override value when stopping at a breakpoint. Continue the NC program by pushing the **NC Start** button.
	Feed rate set to 0%, press and turn to resume When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by pushing the **NC Start** button and increasing the override value.

Icon or button	Meaning
	**Feed rate set to 0%, turn to resume** When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by increasing the override value. Refer to your machine manual. The machine manufacturer uses the optional machine parameter **resumeByTurning** (no. 141801) to define if this button is available.
Perform conditional stop	Toggle switch for activating and deactivating breakpoints **Further information:** "Breakpoints", Page 541

The following functions are available also without the override controller:

- **Feed rate F LIMIT**
  Further information: "Feed rate limit F LIMIT", Page 417
- **Skip block**
  Further information: Programming and Testing User's Manual
- **Pause at M1**
  Further information: Programming and Testing User's Manual

Breakpoints

The control offers the following breakpoints:

Breakpoint	Meaning
Before switch to rapid traverse	The control stops at each change from the feed rate **F** to rapid traverse **FMAX**.
Before switch to feed rate	The control stops at each change from rapid traverse **FMAX** to the feed rate **F**.
Between two rapid traverses	The control stops between two directly sequential **FMAX** rapid traverse movements.
Before tool call	The control stops before every physical tool call with **TOOL CALL**. The control does not stop, for example, before a **TOOL CALL** that simply changes the spindle speed.
Before tilting the working plane	The control stops before NC blocks with the following syntax elements: ■ **PLANE** functions (#8 / #1-01-1) ■ **M128** (#9 / #4-01-1) ■ **FUNCTION TCPM** (#9 / #4-01-1) ■ Cycle **19 WORKING PLANE** (#8 / #1-01-1) You can still execute NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Breakpoint	Meaning
Before cycle call	The control stops before NC blocks with the following syntax elements: ■ **M89** The control stops before each machining position. ■ **M99** ■ **CYCL CALL** ■ **CYCL CALL POS** ■ **CYCL CALL PAT** The control stops before each machining position. ■ Cycles **220 POLAR PATTERN**, **221 CARTESIAN PATTERN**, **224 DATAMATRIX CODE PATTERN** The control stops before each machining position.
In cycle call	**Stop before the first infeed** In the cycles below, the control stops before the first infeed: ■ Cycles for drilling and thread machining ■ Cycles for cylinder surface machining (#8 / #1-01-1) ■ Cycles for Grinding (#156 / #4-04-1) (#156 / #4-04-1) **Further information:** User's Manual for Machining Cycles
	Stop before every infeed In the cycles below, the control stops before every infeed: ■ Milling cycles ■ Mill-turning cycles (#50 / #4-03-1) **Further information:** User's Manual for Machining Cycles
	No stop The control will not stop in the following cycles: ■ Programmable touch probe cycles **Further information:** Measuring Cycles for Workpieces and Tools User's Manual ■ Cycles for monitoring **Further information:** User's Manual for Machining Cycles

The control displays active breakpoints on the **PGM** tab of the **Status** workspace.
Further information: "The PGM tab", Page 135

Displaying breakpoints

The control displays breakpoints with the following icons:

Icon	Meaning
	Active stop The control has detected a breakpoint and stops program run or the simulation at this point.
	Inactive stop The control has detected a breakpoint but does not stop program run or the simulation at this point. In order to stop before this NC block, you must first activate the corresponding toggle switch in the **Program run options** window. **Further information:** "The Program run options window", Page 540

The control displays the icons for breakpoints in the NC program before the block number as soon as at least one conditional stop is active in the **Program run options** window.

When you select an icon, the control displays the name of the associated breakpoint.

Notes

- The override controller is also effective as a feed rate and/or rapid traverse override in the **Manual** operating mode.
- If the NC program contains breakpoints, the control displays a check mark in the **Perform conditional stop** area of the **Tool check** column.

  Further information: "The Tool check column in the Program workspace", Page 209
- If you turn the override controller down with a sudden jerk, the control will automatically set the feed-rate override to 0%, even if the controller itself did not reach 0%.

  The control will resume the NC program if you turn the override controller up again. You don't need to press the **NC Start** key for this. This behavior is independent of the settings in the **Program run options** window.

  After turning the controller down with a sudden jerk, must turn the feed rate back up to 100% from 0%.
- When the execution cursor reaches a breakpoint, the two icons overlap so you can see why the control stops.
- If the **Feed rate set to 0%, turn to resume** button is active, the control reacts as follows:
  - You can continue the NC program only following a conditional stop and by increasing the override value. Otherwise an **NC Start** is necessary (e.g., when starting a program).
  - When the NC program includes two subsequent conditional stops, the 0% override value cannot be changed for 0.3 seconds. This way, the control ensures that you will not continue beyond both conditional stops by just one movement of the override controller.
  - After a conditional stop with a manual tool change you must press the **NC Start** button. You can't continue the NC program by increasing the override value.

Notes about machine parameters

Refer to your machine manual.

- The machine manufacturer defines the maximum override value for rapid traverse. If the maximum override value is, for example, 100% and you enter a rapid-traverse override value greater than 100%, the control still calculates with 100%. If you turn the dial down in this case, then there is no immediate effect. Only once the override controller actually reaches 100% will the control change the override value.
- The machine manufacturer can use the optional machine parameter **ocWaitTime** (no. 103412) to define whether a waiting time will be effective in the cases below:
  - When the program is continued at 0 % after a breakpoint
  - When 100% of the override value is reached

22

Embedded Workspace and Extended Workspace

22.1 Embedded Workspace (#133 / #3-01-1)

Application

You use Embedded Workspace to operate a Windows PC and display its screen contents on the control's user interface. You use Remote Desktop Manager to connect the Windows PC (#133 / #3-01-1).

Related topics

- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608
- Using Extended Workspace to operate a Windows PC through an additional connected monitor
  Further information: "Extended Workspace", Page 548

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (#133 / #3-01-1)
- Connection defined in the machine parameter **CfgRemoteDesktop** (no. 133500)
  In the optional machine parameter **connections** (no. 133501), the machine manufacturer enters the name of the RemoteFX connection.
  Refer to your machine manual.

Description of function

Embedded Workspace is available on the control as an operating mode and as a workspace. If the machine manufacturer does not define a name, then the operating mode and workspace are both named **RDP**.

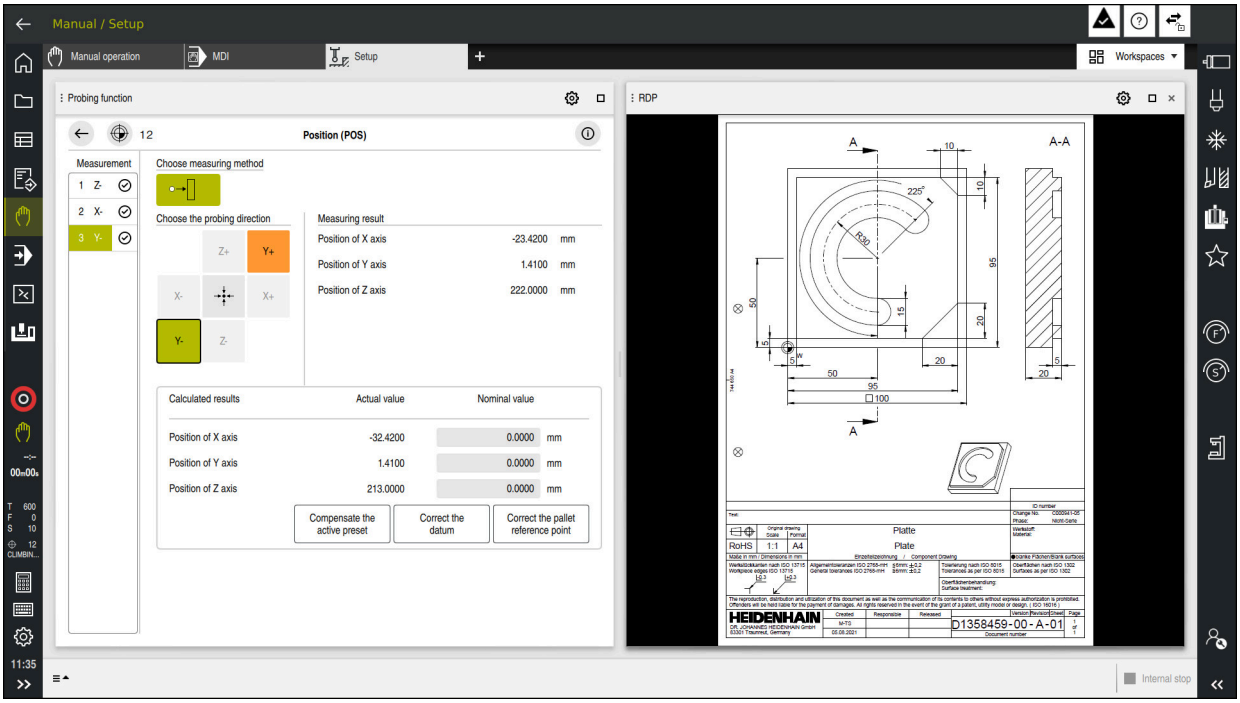
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 609

If you open Embedded Workspace as an operating mode, the control displays a full-screen version of the Windows PC user interface in it.

If you open Embedded Workspace as a workspace, you can change the size and position of the workspace as you wish. The control rescales the user interface of the Windows PC after each modification.

Further information: "Workspaces", Page 85



Embedded Workspace as workspace with opened PDF file

The RDP settings window

If Embedded Workspace is open as a workspace, you can open the **RDP settings** window.

The **RDP settings** window contains the following buttons:

Button	Meaning
Reconnect	If the control could not establish a connection to the Windows PC, for example due to a timeout, press this button to try again. The control can also display this button in the operating mode and the workspace.
Adjust resolution	With this button the control rescales the user interface of the Windows PC to the size of the workspace.

22.2 Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

- Using Embedded Workspace to operate a Windows PC within the control's user interface (#133 / #3-01-1)

  Further information: "Embedded Workspace (#133 / #3-01-1)", Page 546
- ITC hardware expansion

  Further information: "Hardware enhancements", Page 80

Requirement

- Additional attached monitor configured by the machine manufacturer as Extended Workspace

  Refer to your machine manual.

Description of function

Here are some functions you can perform with Extended Workspace:

- Opening files from the control (e.g., drawings)
- Opening windows from HEROS functions in addition to the control's user interface

  Further information: "HEROS menu", Page 663
- Displaying and operating computers connected through Remote Desktop Manager (#133 / #3-01-1)

  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

23

Integrated functional safety (FS)

Application

The safety design of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety design automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety design.

Requirements

- On controls with **SIK**:
  - Software option 160 (Integrated FS: Basic) or software option 161 (Integrated FS: Full)
  - Software options 162 to 166 (Add. FS Ctrl. Loop) or software option 169 (Add. FS Full) as needed

    Whether you need these software options depends on the machine's number of motors.
- On controls with **SIK2**:
  - Integrated FS: Basic software option (#6-30-1)
  - Integrated FS: Full software option as needed (#6-30-2*)

    If your control is equipped with **SIK2**, software option #6-30-1 will enable four safe axes. You can order software option #6-30-2* multiple times and thus enable up to six additional safe axes.
- The machine manufacturer must adapt the FS safety design to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g., guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "The Axis status menu item", Page 553

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of motors using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the motors
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the motors
SLS	Safely Limited Speed	Safely limited speed. Prevents the motors from exceeding the specified speed limits when the guard door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:

Refer to your machine manual.

The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM 1	Operating mode **SOM_1**	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode **SOM_2**	Safe operating mode 2: Setup mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified users
SOM 4	Operating mode **SOM_4** This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

On a control with functional safety (FS), the monitored operating states of the speed **S** and feed rate **F** are displayed in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

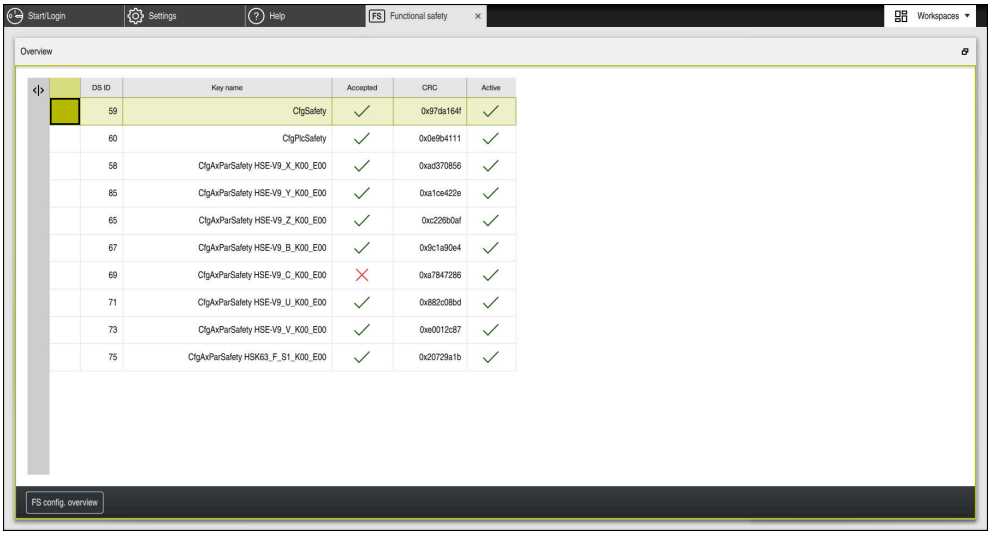
Further information: "Axis display and position display", Page 120

The Functional safety application

Refer to your machine manual.

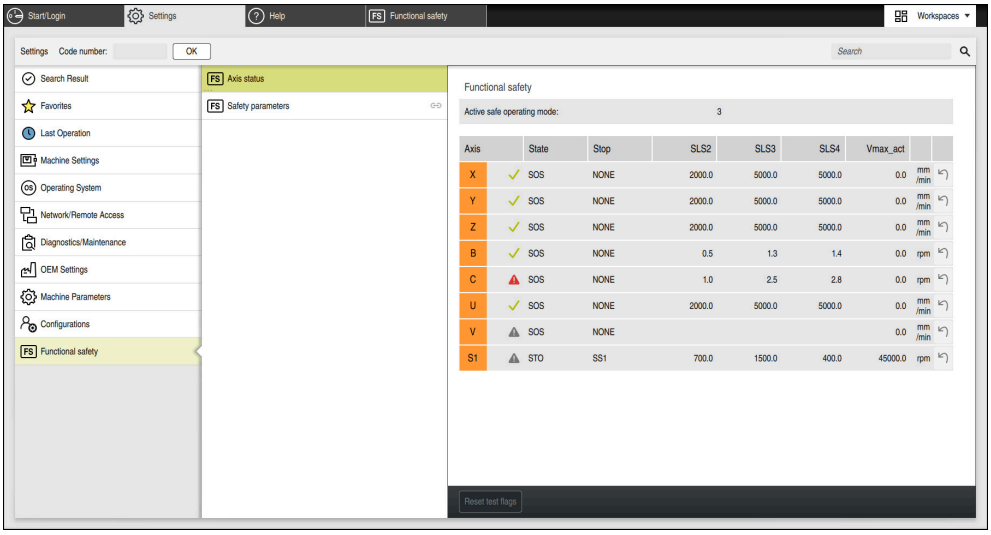
The machine manufacturer configures the safety functions in this application.

In the **Functional safety** application in the **Home** operating mode, the control provides information about the status of the individual safety functions. In this application you can see whether individual safety functions are active and have been accepted by the control.



The **Overview** workspace in the **Functional safety** application

The Axis status menu item



The **Axis status** menu item in the **Settings** application

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction **Further information:** "Functional safety (FS) in the Positions workspace", Page 552
SLS2	Maximum speed or feed-rate values for **SLS** in the **SOM_2** operating mode
SLS3	Maximum speed or feed-rate values for **SLS** in the **SOM_3** operating mode
SLS4	Maximum speed or feed-rate values for **SLS** in the **SOM_4** operating mode This function must be enabled and adapted by the machine manufacturer.
Vmax_act	Currently valid speed or feed-rate limit These are either values from the **SLS** settings or from the SPLC If values are greater than 999 999, the control displays **MAX**.

The control shows the following icons and buttons:

Icon or button	Meaning
↶	Reset the test status of the selected axis
Reset test flags	Reset the test statuses of all axes

- Resetting of the test status is a function for the Service department. Use this function only if instructed by HEIDENHAIN or the machine manufacturer.
- To reset the test status of axes, you need the NC.ApproveFsAxis right. This right is only available if user administration is active.

  Further information: "User administration", Page 635

  Further information: "User administration roles and rights", Page 687

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 555

The control indicates the test status of the individual axes with the following icons:

Icon	Meaning
✓	The axis has been tested or does not need to be tested.
⚠	The axis has not been tested, but must be tested to ensure safe operation. **Further information:** "Checking axis positions manually", Page 555
⚠	The axis is not monitored by functional safety (FS) or is not configured as a safe axis. The axis is monitored by functional safety (FS), but the **SLP** safety function is deactivated. In machine parameter **safeAbsPosition** (no. 403130), the machine manufacturer defines whether the **SLP** safety function is activated for an axis.

23.1 Checking axis positions manually

Refer to your machine manual.
This function must be adapted by your machine manufacturer.
The machine manufacturer defines the test position.

To check the position of an axis:

- Select the **Manual** operating mode
- Select **Move to ref. point**
- Select **Check axis positions** in the **Referencing** workspace
- Select the desired axis
- Press the **NC start** key
- > The axis moves to the test position.
- > After the test position has been reached, the control issues a message.
- Press the **permissive button** on the machine operating panel
- > The control displays the axis as a tested axis.

NOTICE

Danger of collision!

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

- If necessary, move to a safe position before approaching the test positions
- Watch out for possible collisions

In the **Referencing** workspace, you can switch as desired between the **Referencing** and **Check axis positions** modes.

Notes

- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The machine manufacturer defines the behavior of speed-controlled FS-NC axes while the guard door is open in the machine parameter **speedPosCompType** (no. 403129). The machine manufacturer can allow, for example, switching-on of the spindle and thus enable scratching of the workpiece while the guard door is open. Refer to your machine manual.
- The control performs repeated self-tests to detect defective cables, for example. The machine manufacturer defines at what intervals the control will perform the self-tests. When a self-test of the control is active, the control displays an icon in the information bar. The control cannot perform any axis movements while a self-test is active.

24

The Settings application

24.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Icon	Menu item
	Machine Settings		**Machine Settings** **Further information:** "The Machine Settings menu item", Page 562
			General Information **Further information:** "The General Information menu item", Page 564
			SIK **Further information:** "The SIK menu item", Page 565
			Machine Times **Further information:** "The Machine Times menu item", Page 568
			Overview of touch probes **Further information:** "Overview of touch probes menu item", Page 569
			Adjustment of analog voltage offset **Further information:** "Adjustment of analog voltage offset menu item", Page 572
			Set Up Wireless Handwheel **Further information:** "Setting up a wireless handwheel ", Page 534
	Operating System		**Date/Time** **Further information:** "The Adjust system time window", Page 573
			Language/Keyboards **Further information:** "Conversational language of the control", Page 574
			About HeROS **Further information:** "Information on licensing and use", Page 74
			SELinux **Further information:** "SELinux security software", Page 576
			UserAdmin **Further information:** "The User administration window", Page 644
			Current User **Further information:** "The Active user window", Page 645
			Touchscreen Configuration You can select the touchscreen sensitivity and define whether touch points should be shown or hidden.

Icon	Category	Icon	Menu item
	Network/Remote Access		**Shares** **Further information:** "Network drives on the control", Page 577
			Network **Further information:** "Ethernet interface", Page 581
			PKI Admin Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "PKI Admin", Page 588
			OPC UA **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590
			DNC **Further information:** "The DNC menu item", Page 598
			Embedded Workspace Show the connection status **Further information:** "Embedded Workspace (#133 / #3-01-1)", Page 546
			Printer **Further information:** "Printers", Page 601
		VNC	**VNC** **Further information:** "The VNC menu item", Page 605
			Remote Desktop Manager **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608
		VNC	**Real VNC Viewer** Connect to a remote device via a VNC server Available only to network specialists
			Firewall **Further information:** "Firewall", Page 614

Icon	Category	Icon	Menu item
	Diagnostics/Maintenance		**Terminal program** Enter and execute console commands
			HeLogging Define settings for internal diagnostic files
			Portscan **Further information:** "Portscan", Page 619
			perf2 Check processor load and process load
			TNCdiag **Further information:** "TNCdiag", Page 623
			TNCscope Available only to authorized specialists
			NC/PLC Backup **Further information:** "Backup and restore", Page 620
			NC/PLC Restore **Further information:** "Backup and restore", Page 620
			Touchscreen Cleaning The control disables the touchscreen for input for 90 seconds.
			Update the documentation **Further information:** "Update the documentation", Page 624
	OEM Settings		Settings for the machine manufacturer
	Machine Parameters		The group contains machine parameters that can be edited, depending on your rights (e.g., **MPs for setters**). **Further information:** "Machine parameters", Page 626
	Configurations		**Configurations** **Further information:** "Configuring the control's user interface", Page 632
FS	**Functional safety**	FS	**Axis status** **Further information:** "The Axis status menu item", Page 553
		FS	**Safety parameters** **Further information:** "The Functional safety application", Page 552

24.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field. This input field is accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Meaning
123	Editing machine-specific user parameters **Further information:** "Machine parameters", Page 626
555343	Special functions for programming with variables **Further information:** Programming and Testing User's Manual Special functions defining the machine behavior **Further information:** Programming and Testing User's Manual
0	Resetting active code numbers

The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

24.3 The Machine Settings menu item

Application

In the **Machine Settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation
  Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Machine Settings

The Unit of Measure area

In the **Unit of Measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10 before entering it.

Channel Settings

The control displays the channel settings separately for the **Editor** and the **Manual** and **Program Run** operating modes.

You can define the following settings:

Setting	Meaning
Active Kinematics	Use the **Active Kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, have been programmed for other machines. The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose. The control displays the active kinematics model in **Machine** mode of the **Simulation** workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage. **Further information:** "Tool usage test", Page 208 You select when the control should generate a tool-usage file: ■ **Never** The control does not generate a tool-usage file. ■ **Once** The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ **Always** When you simulate or run an NC program, the control will generate a tool-usage file each time.

Traverse Limits

Use the **Traverse Limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse Limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents **Valid** or **Invalid**.
Lower Limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper Limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: Programming and Testing User's Manual

Notes

NOTICE

Danger of collision!

You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!

- ▶ Use the **Active Kinematics** function for the simulation only
- ▶ Use the **Active Kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active Kinematics** function can be selected.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- You can open the tool-usage file in the **Tables** operating mode.
  Further information: "Tool usage file", Page 497
- If the control generated a tool-usage file for an NC program, the **T usage order** and **Tooling list** tables contain data (#93 / #2-03-1).
  Further information: "T usage order (#93 / #2-03-1)", Page 500
  Further information: "Tooling list (#93 / #2-03-1)", Page 502

24.4 The General Information menu item

Application

In the **General Information** menu item of the **Settings** application, the control provides information about the control and the machine.

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **General Information**

The Version Information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ **Control Model** Designation of the control (managed by HEIDENHAIN) ■ **NC-SW** Number of the NC software (managed by HEIDENHAIN) ■ **NCK** Number of the NC software (managed by HEIDENHAIN)
PLC	**PLC-SW** Number or name of the PLC software (managed by the machine manufacturer)

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

The Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 592

The Machine information area

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 592

24.5 The SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control
  Further information: "Software options", Page 67

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **SIK**

The SIK Information area

The control displays the following information:

- **Serial Number**
- **ID number**
- **Control Model**
- **Performance Class**
- **Features**
- **Status**
  The control shows whether it is equipped with a **SIK** or **SIK2**.
- **Temporarily enable options / Disable options**

The Machine manufacturer key area

In the **Machine manufacturer key** area, the machine manufacturer can define a manufacturer-specific password for the control.

The General key area

In the **General key** area the machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing).

The control indicates the status of the general key:

Status	Meaning
NONE	The general key has not yet been used for this software version.
dd.mm.yyyy	Date up to which all software options will be available. Once the general key has expired, it cannot be used again.
EXPIRED	The general key has expired for this software version.

If the software version of the control is increased (e.g., by an update), then the **General key** can be used again.

The Software Options area

In the **Software Options** area, the control shows all available software options in a table.

Column	Meaning
#	Number of the software option
Option	Name of the software option On controls with **SIK2**, the part number and the name of the software option are displayed.
	The control indicates the status of the software option by means of the following symbols: ■ No symbol: The software option is not enabled. ■ Checkmark: The software option is enabled permanently with all functions. ■ Clock symbol: The software option has been enabled for a limited period of time or can be ordered again on controls with **SIK2**. ■ Padlock: The software option has been locked by the machine manufacturer.
Expiration Date or **Status**	The control displays the following information on the status of the software option: ■ **Enabled** ■ **YYYY-MM-DD** If a software option has been enabled for a limited period of time, the control shows the date up to which it will be available. ■ **X of X** On controls with **SIK2**, the control shows how often the software option has been enabled.
Details	Detailed information for the machine manufacturer
Config.	Function that the machine manufacturer can use to lock software options

24.5.1 Viewing of software options

To view enabled software options on the control:

- ▶ Select the **Home** operating mode
- ▶ Select the **Settings** application
- ▶ Select **Machine Settings**
- ▶ Select **SIK**
- ▶ Navigate to the **Software Options** area
- > For enabled software options, the control displays the text **Enabled**.

Definition

Abbreviation	Definition
SIK (System Identification Key)	**SIK** is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the **SIK**. The software options are saved on the **SIK**. 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.

24.6 The Machine Times menu item

Application

In the **Machine Times** menu item of the **Settings** application, the control shows the run times since being put into service.

Related topics

- Date and time of the control
  Further information: "The Adjust system time window", Page 573

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **Machine Times**

The control displays the following machine times:

Machine time	Meaning
Control On	Run time of the control since being put into service
Machine On	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service

Refer to your machine manual.
The machine manufacturer can define up to 20 additional run times.

24.7 Overview of touch probes menu item

Application

The **Overview of touch probes** menu item of the **Settings** application allows you to create and manage all workpiece touch probes and tool touch probes of the control.

Related topics

- Touch probe table
  Further information: "Touch probe table tchprobe.tp", Page 489
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)
  Further information: "Machine parameters", Page 626

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Overview of touch probes

The control displays one table for the **SE** transceivers, one table for the **TS** workpiece touch probes, and one table for the **TT** tool touch probes.

The tables contain the following information:

- Model
- Number
  Only for **TS** and **TT**
- Serial number
- **Add** button
  Only for **TS** and **TT**

The control shows the table for transceivers only if you use touch probes with radio transmission.

Transceiver unit

If you use touch probes with radio transmission, the control displays the following information for **Transceiver unit**:

Display	Meaning
Status	The transceiver unit is active or inactive
SE	Select the **SE** transceiver
Dimension	Select or change the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or handwheels with radio transmission.

Details

If you use touch probes with radio transmission, the control displays the following information for **Details**:

Display	Meaning
Signal strength	The signal strength in the bar chart The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision detected
Battery status	If the battery charge falls below the plotted limit, the control displays a warning.

For touch probes with infrared transmission, the control displays the following information for **Details**:

There are no further diagnostic data and functions available for this touch probe.

Buttons

The control displays the following buttons:

Button	Meaning
+	**Add** The control adds a new row to the respective table. You define a workpiece touch probe in the Touch probe table and a tool touch probe in the machine parameter **CfgTT** (no. 122700).
Connect	Connect a radio touch probe to a transceiver
Switch-on	Switch touch probe on
Switch off	Switch touch probe off
Change	Change the radio channel used for communication by the touch probe and the transceiver
TNCdiag	The control opens TNCdiag. **Further information:** "TNCdiag", Page 623
Touch probe table	The control opens the Touch probe table.
Delete	The control deletes the highlighted table row.

24.7.1 Connecting a new touch probe with radio transmission

To connect a touch probe with radio transmission:

- Select the **Settings** application
- Select **Machine Settings**
- Select **Overview of touch probes**
- Select the desired transceiver unit (e.g., SE 661)
- Select **Connect new touch probe**
- > The control opens the **Connecting... Please insert the batteries in the touch probe** window.
- Insert the battery into the touch probe
- > The control closes the window once the touch probe has been connected.
- Select the newly connected touch probe
- Select **Switch-on**
- > The status of the touch probe changes to **On**.

24.7.2 Changing the radio channel

Before changing the radio channel, ensure that the touch probe is switched off and that the desired transceiver unit is not connected to any active touch probe.

To change the radio channel:

- Select the **Settings** application
- Select **Machine Settings**
- Select **Overview of touch probes**
- Select the desired transceiver unit (e.g., SE 661)
- Select **Change**
- > The control opens the **Change** window.
- Select a new channel number
- > The control closes the window and displays the new channel number.

24.8 Adjustment of analog voltage offset menu item

Application

The **Adjustment of analog voltage offset** menu item of the **Settings** application allows you to correct the voltage offsets of all analog axes that are present.

Requirement

- Machine with analog axes
- If user administration is active, the NC.Setter role
  Further information: "List of roles", Page 687

Description of function

To navigate to this function:
Settings ▶ **Machine Settings** ▶ **Adjustment of analog voltage offset**

Column	Meaning
Axis	All analog axes
Axis is in a closed control loop	Axis is in a closed control loop or is not in a closed control loop. If the axis is in a closed control loop, the control shows a green checkmark.
Offset in mV	Voltage offset If permitted by the axis configuration, you can adjust the voltage offset here.

If you adjust the voltage offsets in the **Offset in mV** column, the control synchronizes the optional machine parameter **analogOffset** (no. 402810) with the current voltage offsets.

24.9 The Adjust system time window

Application

In the **Adjust system time** window, you can set the time zone, date and time manually or by means of NTP server synchronization.

Related topics

- Run times of the machine tool
  Further information: "The Machine Times menu item", Page 568

Description of function

To navigate to this function:

Settings ▶ **Operating System** ▶ **Date/Time**

The **Adjust system time** window contains the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: ■ Year ■ Month ■ Day ■ Time
Synchronize the time over NTP server	If you activate this check box, the control will automatically synchronize the system time with the defined NTP server. You can add a server with a host name or a URL.
Time zone	You can select your time zone from a list.

24.10 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control
  Further information: "Machine parameters", Page 626

Description of function

To navigate to this function:

Settings ▶ Operating System ▶ Language/Keyboards

You can't define two different conversational languages for the operating system and control.

The **helocale** window consists of the following areas:

Area	Function
Language	Choose the HEROS conversational language from a selection menu Only if the machine parameter **applyCfgLanguage** (no. 101305) is defined as **FALSE**.
Keyboards	Select the language layout of the keyboard for HEROS functions

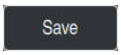
24.10.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

- Select the **Settings** application
- Enter the code number 123
- Select **OK**
- Select **Machine Parameters**
- Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- Navigate to the machine parameter **ncLanguage** (no. 101301)
- Select the desired language

- Select **Save**
- > The control opens the **Configuration data changed. All changes.** window.



- Select **Save**
- > The control opens the notification menu and displays a "Question type" error.

- Select **CLOSE CONTROL**
- > The control restarts.
- > Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.

Notes

- Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.
  - **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

    Further information: "Changing the language", Page 575
  - **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.
- Use the optional machine parameter **noRebootDialog** (no. 101306) to define whether, after a change of the conversational language, the control displays a restart message.

24.11 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions (such as viruses and other malicious software).

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall
  Further information: "Firewall", Page 614

Description of function

To navigate to this function:

Settings ▶ **Operating System** ▶ **SELinux**

By default, **SELinux** access control is implemented as follows:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Safety-relevant files, such as **SELinux** system files or HEROS boot files, may only be modified using explicitly selected programs.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
  - Software update: A software update from HEIDENHAIN can replace or modify system files.
  - SELinux configuration: The configuration of **SELinux** in the **Security Policy Configuration** window is usually protected by a password defined by the machine manufacturer. Please refer to the machine manual.

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. **SELinux** is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of **SELinux** permit it.

24.12 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management
  Further information: Programming and Testing User's Manual
- Network settings
  Further information: "Ethernet interface", Page 581

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **Shares**

You can define any number of network drives, but only seven can be connected at a time.

The Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the **Mount** column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the **Auto** column if an active automatic connection exists.
Add	Define a new connection **Further information:** "The Mount assistant window", Page 579
Remove	Delete an existing connection
Copy	Copy connection **Further information:** "The Mount assistant window", Page 579
Edit	Edit the connection settings **Further information:** "The Mount assistant window", Page 579
Private network drive	User-specific connection if user administration is active The control selects the check box in the **Privat** column if a user-specific connection exists.

The Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

The Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ **Drive name:** Network drive name in the file management of the control The names must be all uppercase letters, terminated by a colon (**:**). ■ **Volume ID:** Currently no function ■ **Private network drive** With user administration active, the connection is only visible to the user who created it.
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address ■ **Share name:** Directory accessed by the control
Automount	**Connect automatically (not possible with the "Ask for password?" option)** The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ **Single Sign On** With user administration active, the control automatically connects an encrypted network drive when the user logs in. ■ **Windows user name:** ■ **Ask for password? (not possible with the "Connect automatically" option)** Select whether a password is required upon connecting. ■ **Password** ■ **Password verification**
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection **Further information:** "Examples of Mounting options", Page 580
Check	The control displays a summary of the defined settings. You can check the settings and save them with **Apply**.

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to include the domain in the user name, but rather specify it as an option.
vers=3.1.1	Protocol version
sec=ntlmssp	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input: **512...8192**
wsize=4096	Packet size in bytes for data transmission Input: **512...8192**
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
nfsvers=2	Protocol version

If you use the CIMCO NFS software, you must enter the option nfsvers = 2. CIMCO NFS supports NFS only up to version 2.

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

24.13 Ethernet interface

Application

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network.

Related topics

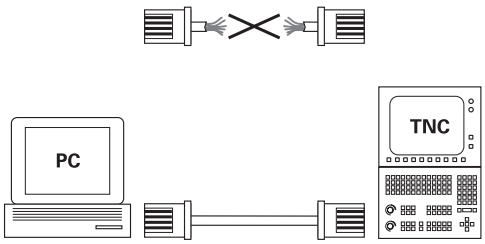
- Firewall settings
  Further information: "Firewall", Page 614
- Network drives on the control
  Further information: "Network drives on the control", Page 577
- External access
  Further information: "The DNC menu item", Page 598

Description of function

The control transfers data via the Ethernet interface using the following protocols:

- **CIFS** (common internet file system) or **SMB** (server message block)
  The control supports versions 2, 2.1 and 3 of these protocols.
- **NFS** (network file system)
  The control supports versions 2 and 3 of this protocol.

Connection options



10BaseT / 100BaseTx / 1000BaseTx

You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	Ethernet connection The control displays the icon at the bottom right in the taskbar. **Further information:** "Taskbar", Page 667 When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions: ■ Connected networks You can disconnect the network connection. Select the network name to reconnect. ■ Available networks ■ VPN connections Currently no function

Notes

- Protect your data and the control by running the machines in a secure network.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

24.13.1 The Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

> Have a network specialist configure the control.

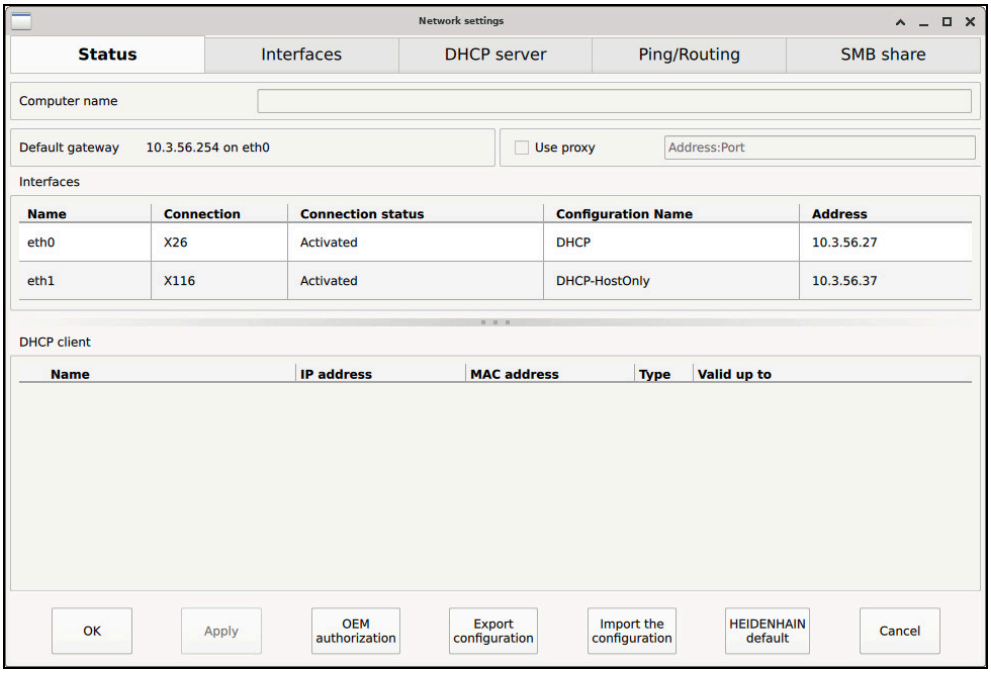
Related topics

- Network configuration
  Further information: "Network configuration with Advanced Network Configuration", Page 681
- Firewall settings
  Further information: "Firewall", Page 614
- Network drives on the control
  Further information: "Network drives on the control", Page 577

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **Network**



The **Network settings** window

The Status tab

The **Status** tab contains the following information and settings:

Area	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name. **Further information:** "Notes", Page 587
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the **address** and the **port** of a proxy server in the network.
Interfaces	The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty. The control displays the following information in the table: ■ **Name** (e.g., **eth0**) ■ **Connection** (e.g., **X26**) ■ **Connection status** (e.g., **CONNECTED**) ■ **Configuration Name** (e.g., **DHCP**) ■ **Address** (e.g., **10.7.113.10**) **Further information:** "The Interfaces tab", Page 585
DHCP client	The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty. The control displays the following information in the table: ■ **Name** Host name and connection status of the device. The control shows the following connection status: ■ Green: Connected ■ Red: No connection ■ **IP address** Dynamically assigned IP address of the device ■ **MAC address** Physical address of the device ■ **Type** Type of connection The control displays the following connection types: ■ **TFTP** ■ **DHCP** ■ **Valid up to** Time until which the IP address is valid without being renewed The machine manufacturer can make settings for these devices. Refer to your machine manual.

The Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.

The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a toggle switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ **CONNECTED** Connected ■ **DISCONNECTED** Connection separated ■ **CONFIGURING** The IP address is being fetched from the server ■ **NOCARRIER** No cable present
Configuration Name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ **DHCP-LAN**: Settings for the standard interface for a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network **Further information:** "Network configuration with Advanced Network Configuration", Page 681 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 681

- If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.
- The control exclusively supports the **Ethernet** connection type.

The DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).

Refer to your machine manual.

The Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Area	Information or Setting
Ping	**Address:Port** and **Address:** You can enter the IP address of the computer and possibly the port number for checking the network connection. Entry: Four numerical values separated by dots and, if necessary, a port number separated by a colon (e.g., **10.7.113.10:22**) As an alternative, you can enter the name of the computer whose connection you want to check. Starting and stopping the test ■ **Start** button: starts the test The control displays status information in the ping field. ■ **Stop** button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

The SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station.

When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.

Further information: Programming station for milling controls
You download the documentation together with the programming station software.

Exporting and importing a network profile

To export a network profile:

- Open the **Network settings** window
- Select **Export configuration**
- > The control opens a window.
- Select the storage location for the network profile (e.g., **TNC:/etc/sysconfig/net**)
- Select **Open**
- Select the desired network profile
- Select **Export**
- > The control saves the network profile.

> You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:

- Open the **Network settings** window
- Select **Import the configuration**
- > The control opens a window.
- Select the storage location of the network profile
- Select **Open**
- Select the desired network profile
- Press **OK**
- > The control opens a window with a prompt.
- Press **OK**
- > The control imports and activates the selected network profile.
- You might need to restart the control

> The **HEIDENHAIN default** button allows you to import the default values of the network settings.

Notes

- Preferably restart the control after making changes in the network settings.
- If you change the computer name of the control, you must reconfigure the LDAP database of user administration.

  Further information: "Local LDAP database", Page 646

  You must also regenerate the server certificates of **OPC UA NC Server** (#56-61 / #3-02-1*).

  Further information: "Login options", Page 592
- The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.

  Further information: "Conversational language of the control", Page 574

24.14 PKI Admin

Application

With **PKI Admin**, you can manage the server and client certificates of **OPC UA NC Server** (#56-61 / #3-02-1*) on the control. To define access rights to the control, you can classify the certificates as trusted or not trusted, for example.

PKI Admin has no functionality without the OPC UA NC Server (#56-61 / #3-02-1*) software option.

Related topics

- Quickly and easily connecting the OPC UA client application to the control (#56-61 / #3-02-1*)

  Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 596

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **PKI Admin**

The **Administration of the PKI Infrastructure** window provides the following tabs:

Tab	Function
Own certificates	The control provides the following areas: ■ **Check the configuration** The control checks the validity of the server certificates. ■ **Use self-created certificate**: ■ **Optional certificate settings** The control adds static IP addresses to the server certificates. You can select the IP address of the **eth0** or **eth1** interface or specify the required IP addresses. ■ **Recreate certificate** The control recreates the server's chain of trust. After the next restart of the control, it will use the new certificate. ■ **Export certificate chain** The control saves the server's chain of trust that you import into the client application. ■ **Use customer-specific certificate**: ■ **Load certificate** You can import a customized certificate. Please note the requirements for self-created certificates for **OPC UA** (#56-61 / #3-02-1*). **Further information:** "Login options", Page 592 ■ **Existing certificates of server** The control displays the available certificates and revocation lists. You can export the selected certificate or the selected revocation list, show its details, or delete it.

Tab	Function
Trusted	The server knows the certificate and trusts it after successful validation. For connection to the server, the client certificate must have been specified on this tab. For a **OPC UA** connection (#56-61 / #3-02-1*), you also need to assign a **OPC UA** license to the certificate. **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 597
Issuers	On this tab, you can specify the issuer of the trusted certificates. The server uses the issuer's information to validate the certificate.
Rejected	On this tab, the control specifies client certificates whose connection attempt to the **OPC UA NC Server** (#56-61 / #3-02-1*) failed. Connection failures can occur in the following situations: ■ The client certificate is unknown and has not been classified as trusted. If you want to connect the client application to the server, you can use the **Move** function to move the certificate to the **Trusted** tab. ■ A trusted client certificate has expired.
Revocation lists	On this tab, you can specify CRL files that list untrusted certificates. The server prohibits connections that use these certificates. In the **Settings for revocation lists** area, you can permit connections of applications with certificates in a multi-level certificate chain even if no associated CRL files exist.

Definition

PKI
PKI (public key infrastructure) is the management structure for digital certificates that are required for secure communication. A digital certificate has the same purpose as an identity card or passport. With a digital certificate, its owner can encrypt, sign and authenticate the communication.

24.15 OPC UA NC Server (#56-61 / #3-02-1*)

24.15.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange across operating systems between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The security analysis was updated in 2022. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Application

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.

Pay attention to the interface documentation of the **OPC UA NC Server** as well as the documentation of the client application.

Related topics

- **Information Model** interface documentation with the specification of the **OPC UA NC Server** in English
  ID: 1309365-xx or **OPC UA NC Server Interface Documentation**
- Quickly and easily connecting the OPC UA client application to the control
  Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 596
- User roles and user rights for OPC UA
  Further information: "User administration roles and rights", Page 687
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 678

Requirements

- OPC UA NC Server (#56-61 / #3-02-1*) software options

  For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).

  If your control features the **SIK2**, you can order this software option multiple times and enable up to ten connections.
- Firewall configured

  Further information: "Firewall", Page 614
- The OPC UA client supports the **security policy** and authentication method of **OPC UA NC Server**:
  - **Security Mode: SignAndEncrypt**
  - **Algorithm:**
    - **Basic256Sha256**
    - **Aes128Sha256RsaOaep**
    - **Aes256Sha256RsaPss**
  - **User authentication:**
    - **X509 certificates**
    - User name and password
- For logon with the user name and password:
  - Permitted by the machine manufacturer
  - User administration is active
  - NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet right

Description of function

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- Creation of service files
- Read and write tool data (the corresponding right is required)
- Read from and write to the counter (the corresponding right is required)
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (the corresponding right is required)
- Validation of 3D models for tool carriers

  Further information: "Tool carrier management", Page 201
- Validate 3D models for tools (#140 / #5-03-2)

  Further information: "Tool model (#140 / #5-03-2)", Page 205

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to query general machine information, such as the year of construction of the machine or its location.

The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)
  Further information: "The Machine information area", Page 564
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)
  Further information: "The Info about machine manufacturer area", Page 564

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives.

The following actions are permitted:

- Creating and deleting folders
- Reading, editing, copying, moving, creating, and deleting files

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. As long as the operating system is active, you can create and transmit service files, for example.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- ▶ System-relevant files must be edited only by authorized specialists

Login options

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them (the application instance certificates) in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions. As an alternative to the user certificate, the **OPC UA NC Server** also permits login with a user name and password.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function.

All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

Further information: "PKI Admin", Page 588

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- ▶ Open the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **SSH keys and certificates**
- ▶ Select **Import certificate**
- > The control opens the **Import certificate** window.
- ▶ Select the certificate
- ▶ Select **Open**
- > The control imports the certificate.
- ▶ Select **Use for OPC UA**
- > The control uses the certificate for **OPC UA**.

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
  - File format: *.der
  - Signature with hash SHA256
  - Validity period of at most 5 years is recommended
- Client certificates
  - Host name of the client
  - Application URI of the client
- Server certificates
  - Host name of the control
  - Application URI of the server according to the following structure: urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
  - Validity period of 20 years maximum

Login with user name and password

The machine manufacturer can permit login with a user name and password, for example for client applications that do not support login with a user certificate.

For this type of login, a user with NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet rights must exist while user administration is active.

In the **OPC UA** menu item of the **Settings** application, the control indicates the options available to the current user for logging in.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 595

Notes

- OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.
- Refer to your machine manual.

  The machine manufacturer can create additional function users (for example, to enable client applications in order to access specific machine data when user administration is active).

  Further information: "User administration", Page 635

24.15.2 The OPC UA (#56-61 / #3-02-1*) menu item

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the **OPC UA NC Server**.

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **OPC UA**

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether the **OPC UA NC Server** is active: ■ Green icon **OPC UA NC Server** is active ■ Gray icon: **OPC UA NC Server** is not active or software option not enabled You can manually start or restart the **OPC UA NC Server** as required. **Further information:** "Manually starting the OPC UA NC Server", Page 595
OPC UA connection assistant	Open the **OPC UA NC Server connection assistant** window **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 596
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 597
PKI Admin	Open the **Administration of the PKI Infrastructure** window **Further information:** "PKI Admin", Page 588
Login options of the current user	The control shows whether the options are available for the current user: ■ User certificate ■ User name and password Possible only if user administration is active
Host computer operation	Activate or deactivate host computer operation with a toggle switch **Further information:** "The DNC area", Page 598

When a connection is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 97

Manually starting the OPC UA NC Server

You can manually start or restart the **OPC UA NC Server** as required. Thus, you can apply changes made to the machine parameters or the certificates, which are relevant to the server, without having to shut down the control.

While an OPC UA connection is active, the control displays a confirmation prompt before the restart. During the restart, the control will disconnect active connections automatically.

For this function, you need the HEROS.SetNetwork permission.

Further information: "User administration roles and rights", Page 687

24.15.3 The OPC UA connection assistant function (#56-61 / #3-02-1*)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection assistant** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning the OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-6 using the **OPC UA NC Server - License Settings** window
  Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 597
- Managing certificates with the **PKI Admin** menu
  Further information: "PKI Admin", Page 588

Description of function

Use the **OPC UA** menu item to open the **OPC UA NC Server connection assistant** window.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 595

The assistant provides the following steps:

1 Export **OPC UA NC Server** certificates
2 Import the certificates of the OPC UA client application
3 Assign each of the available OPC UA NC Server software options to a OPC UA client application
4 Select the type of user logon: certificate or password
  If you select the user logon with a password, the control will skip the following steps up to the firewall.
5 Import the user certificates
6 Assign the user certificates to users
7 Configure the firewall
8 Connect the client application to the control

If at least one software option is active for the OPC UA NC Server, the control will generate the server certificate as a part of a self-generated certificate chain during the first start-up. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. The user name and the password are defined in the user administration. Please contact your IT department.

Note

The **OPC UA NC Server connection assistant** also helps you create test or sample certificates for users and the OPC UA client application. Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

24.15.4 The OPC UA license settings function (#56-61 / #3-02-1*)

Application

You can use the **OPC UA NC Server - License Settings** window to assign an OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-6.

Related topics

- Setting up the OPC UA client application with the **OPC UA connection assistant** function

  Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 596
- Managing certificates with **PKI Admin**

  Further information: "PKI Admin", Page 588

Requirement

- Certificate has been added to the **Trusted** category in **PKI Admin**

Description of function

Use the **OPC UA** menu item to open the **OPC UA license settings** window.

After using the **OPC UA connection assistant** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you enable the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

24.16 The DNC menu item

Application

The **DNC** menu item allows you to permit or block access to the control (e.g., connections via a network or TNCremo).

Related topics

- Connecting network drives
  Further information: "Network drives on the control", Page 577
- Setting up a network
  Further information: "Ethernet interface", Page 581
- TNCremo
  Further information: "PC software for data transfer", Page 671
- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **DNC**

The **DNC** area contains the following icons:

Icon	Meaning
	**Add** a computer-specific connection
	**Edit** a computer-specific connection
	**Delete** a computer-specific connection

When a connection is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 97

The DNC area

In the **DNC** area you use toggle switches to activate the following functions:

Switch	Meaning
DNC access permitted	Permit or block all accesses to the control through a network
TNCopt full access allowed	Permit or block full access via TNCopt Only for testing
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation If host computer operation is active, the control displays the **Host computer is active** message in the information bar. You cannot use the **Manual** and **Program Run** operating modes. You cannot activate host computer operation while running an NC program.

Secure connections

The control displays general and customized settings for **Secure connections**. You can activate the following functions:

Row	Meaning
Fingerprint of the host key	Pressing the **Show** button tells the control to show a unique ASCII image that is equivalent to a fingerprint. When setting up a secure connection, you can compare this ASCII image to an image within the client application. That way you can ensure that you are connecting to the right control.
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user. Activate this toggle switch only while you are setting up a connection.
Key management	In this row, you open the **Certificate and keys** window. **Further information:** "SSH-secured DNC connection", Page 658

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	■ **Permit** The control permits network access without confirmation. ■ **Inquire** The control asks for confirmation upon a network access attempt. You can choose whether to permit or block the access once or always. ■ **Deny** The control does not permit network access.
Type	■ **Com1** Serial interface 1 ■ **Com2** Serial interface 2 ■ **Ethernet** Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

24.17 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing
  Further information: Programming and Testing User's Manual

Requirement

- PostScript-capable printer
  The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting the PostScript emulation in the printer menu.
  Further information: "Note", Page 604

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Printer ▶ Heros Printer Manager

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: Programming and Testing User's Manual

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 604

There are various methods to start printing:

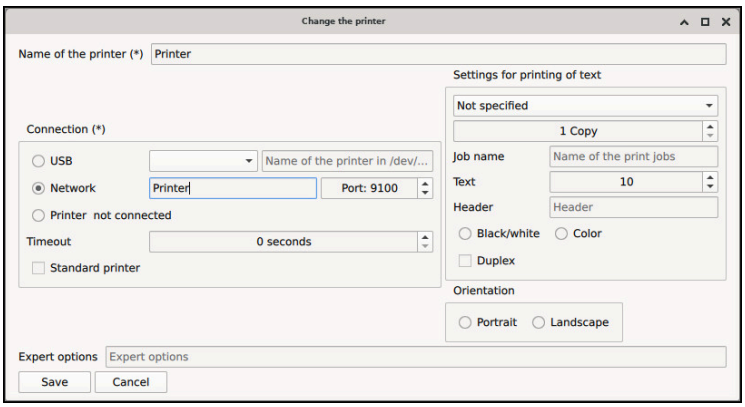
- Copying the file to be printed to the **PRINTER** drive
  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.
  You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.
- Using the **FN 16: F-PRINT** function

Icons and buttons

The **Heros Printer Manager** window provides the following icons and buttons:

Icon or button	Meaning
	**Print a test page** Prints a test page on the selected printer
	**Delete** Deletes the selected printer
CREATE	Creates a printer
Copy	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
Status	Displays the status information of the selected printer

The Change the printer window



You open the window by double-tapping or double-clicking the desired printer.
For each printer, the following properties can be set:

Area	Meaning
Name of the printer	Customizes the printer name
Connection	■ **USB**: The control automatically displays the name ■ **Network**: Network name or IP address of the printer Port for the network printer (default: 9001) ■ **Printer %1 not connected** ■ **Timeout** Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**. Use this setting if the file to be printed is populated with FN functions (e.g., when probing). ■ **Standard printer** Select the default printer The control automatically assigns this setting to the first printer added.
Settings for printing of text	■ Paper size ■ Number of copies ■ **Job name** ■ **Font size** ■ **Header** ■ Printing options ■ **Black/white** ■ **Color** ■ **Duplex**
Orientation	■ **Portrait** ■ **Landscape**
Expert options	Available only to authorized specialists

24.17.1 Creating a printer

To create a new printer:

- Enter the printer name in the input field
- Select **CREATE**
- > The control creates a new printer.
- Double-tap or double-click the printer
- > The control opens the **Change the printer** window.
- Define the properties
- Select **Save**
- > The control applies the settings and displays the defined printer in the list.

24.17.2 Copying a printer

To copy the printer properties of an available printer:

- Select the desired printer
- Enter the name of the new printer in the input field
- Select **Copy**
- > The control creates a new printer with the settings of the selected printer.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

24.18 The VNC menu item

Application

VNC is software that allows you to access the control from remote devices, such as from an additional ITC operating station. **VNC** enables you to transmit screen contents, mouse movements, and keystrokes between devices.

Related topics

- Firewall settings
  Further information: "Firewall", Page 614
- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

Description of function

To navigate to this function:

Settings ▶ **Network/Remote Access** ▶ **VNC**

When an ITC is connected and **Enabling VNC focus** is set, the control displays an icon.

Further information: "The VNC Focus Settings area", Page 607

Further information: "Icons on the control's user interface", Page 97

Icons and buttons

The **VNC settings** window provides the following buttons:

Button	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set standard values	Reset the settings to their default values
Set preferred owner of the focus	Select the **Preferred owner of the focus** check box

The VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.

The control displays the following contents:

Column	Contents
Computer name	IP address or computer name
VNC	Connection of the client to the VNC viewer
VNC Focus	The client participates in the focus assignment
Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect. ■ Enable TeleService and IPC Client via a TeleService connection ■ DHCP Other computer that retrieves an IP address from this computer.

The Firewall warning area

If the firewall blocks **VNC**, the control displays the **Firewall warning** area.

Further information: "Firewall", Page 614

The Global settings area

In the **Global settings** area, you can define the following settings:

Function	Meaning
Enable RemoteAccess and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

The Enabling other VNC area

In the **Enabling other VNC** area, you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

The VNC Focus Settings area

In the **VNC Focus Settings** area, you can define the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated if the focus changes. Only if the **Enabling VNC focus** check box is enabled
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is enabled
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds). This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is enabled

Select the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose (e.g., ITC industrial computers).

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.

  Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	**VNC** is software that allows you to control other devices over a network connection.

24.19 The Remote Desktop Manager window (#133 / #3-01-1)

Application

With Remote Desktop Manager you can display external computer units on the control screen that are connected via Ethernet, and operate them through the control. You can also shut down a Windows computer together with the control.

Related topics

- External access
  Further information: "The DNC menu item", Page 598

Requirements

- Remote Desk. Manager (#133 / #3-01-1) software option
- Existing network connection
  Further information: "Ethernet interface", Page 581

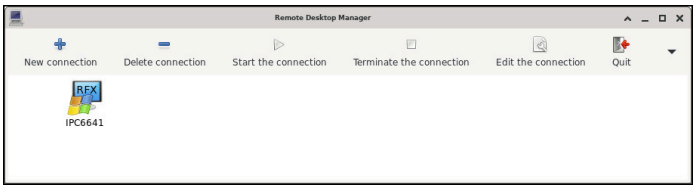
Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Remote Desktop Manager

Remote Desktop Manager grants the following connection options:

- **Windows Terminal Service (RemoteFX)**: Display the desktop of an external Windows computer on the control
  Further information: "Windows Terminal Service (RemoteFX)", Page 609
- **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control
  Further information: "VNC", Page 609
- **Switch-off/restart of a computer**: Automatically shut down a Windows computer together with the control
- **World Wide Web**: Available only to authorized specialists
- **SSH**: Available only to authorized specialists
- **User-defined connection**: Available only to authorized specialists



HEIDENHAIN offers the IPC 6641 as a Windows computer. With the IPC 6641 you can start and operate Windows-based applications directly from within the control.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the operating system is shut down, the control automatically terminates all connections. Please note that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Buttons

Remote Desktop Manager contains the following buttons:

Button	Function
New connection	Create a new connection in the **Edit the connection** window **Further information:** "Establishing and starting a connection", Page 612
Delete connection	Delete the selected connection
Start the connection	Start the selected connection **Further information:** "Establishing and starting a connection", Page 612
Terminate the connection	Terminate the selected connection
Edit the connection	Edit the selected connection in the **Edit the connection** window **Further information:** "Connection settings", Page 610
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection **Further information:** "Exporting and importing connections", Page 613
Export the connections	Back-up the selected connection **Further information:** "Exporting and importing connections", Page 613

Windows Terminal Service (RemoteFX)

You don't need any additional software on a computer for a RemoteFX connection, but you might need to change some settings on the computer.

Further information: "Configuring an external computer for Windows Terminal Service (RemoteFX)", Page 612

For integrating the IPC 6641, HEIDENHAIN recommends using a RemoteFX connection.

With RemoteFX, a separate window opens for the screen of the external computer. The active desktop on the external computer is then locked and the user logged off. This prevents two users from accessing the control simultaneously.

VNC

You need an additional **VNC** server for your external computer when connecting through VNC. Install and configure the VNC server (e.g., TightVNC server) before establishing the connection.

VNC mirrors the screen of the external computer. The active desktop on the external computer is not locked automatically.

With a **VNC** connection you can shut down the external computer through the Windows menu. The computer cannot be restarted through the connection.

Connection settings

General settings

The following settings apply to all connection options:

Setting	Meaning	Usage
Connection name	Name of the connection in **Remote Desktop Manager** You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _	Required
Restarting after end of connection	Behavior after disconnection: ■ **Always restart** ■ **Never restart** ■ **Always after an error** ■ **Ask after an error**	Required
Automatic starting upon login	Connect automatically when starting	Required
Add to favorites	The control displays the connection's icon in the taskbar. Tap or click the icon to start the connection directly.	Required
Move to the following workspace	Number of the desktop for the connection; desktops 0 and 1 are reserved for the NC software. Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can be seen and used only by its creator	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the **IPC6641.machine.net** setting for the IPC 6641. The host name **IPC6641** must be assigned to the IPC in the Windows operating system for this setting.	Required
Password	Password of the user	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

Additional settings for Windows Terminal Service (RemoteFX)

The control offers the following additional connection settings for the **Windows Terminal Service (RemoteFX)** option:

Setting	Meaning	Usage
User name	Name of the user	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or **User-defined window size**	Size of the connection window on the control	Required

Additional settings for VNC

The control offers the following additional connection settings for the **VNC** option:

Setting	Meaning	Usage
Full-screen mode or **User-defined window size:**	Size of the connection window on the control	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated.	Required

Additional settings for Switch-off/restart of a computer

The control offers the following additional connection settings for the **Switch-off/ restart of a computer** option:

Setting	Meaning	Usage
User name	User name with which the connection should log in.	Required
Windows domain:	If required, domain of the target computer	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the **Now you can switch off.** message, it waits for the number of seconds defined here. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the defined number of seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If **Force** is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Restart the windows computer	Required
Run during restart	When the control restarts, restart the Windows computer as well. Effective only if the control is restarted using the shutdown icon at the bottom right in the taskbar or if it is restarted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	Shut down the Windows computer (no restart) when shutting down the control. This is the default behavior. Even the **END** key will then not trigger a restart.	Required

24.19.1 Configuring an external computer for Windows Terminal Service (RemoteFX)

To configure the external computer (e.g., in Windows 10 operating systems):

- Press the Windows key
- Select **Control Panel**
- Select **System and Security**
- Select **System**
- Select **Remote Settings**

> The computer opens a pop-up window.

- Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- In the **Remote Desktop** area, enable **Allow Remote connections to this computer**
- Press **OK** to confirm your settings

24.19.2 Establishing and starting a connection

To establish and start a connection:

- Open **Remote Desktop Manager**
- Select **New connection**

> The control displays a selection menu.

- Select a connection option
- Under **Windows Terminal Service (RemoteFX)**, select the operating system

> The control opens the **Edit the connection** window.

- Define the connection settings
  Further information: "Connection settings", Page 610
- Press **OK**

> The control saves the settings and closes the window.

- Select connection
- Select **Start the connection**

> The control starts the connection.

24.19.3 Exporting and importing connections

To export a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the desired connection
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Export the connections**
- > The control opens the **Select export file** window.
- ▶ Define the name of the saved file
- ▶ Select the target file
- ▶ Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Import connections**
- > The control opens the **Select file to import** window.
- ▶ Select file
- ▶ Select **Open**
- > The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!

If you do not shut down external computers properly, data may be irreversibly damaged or deleted.

- ▶ Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.

  With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

24.20 Firewall

Application

The control provides a firewall to allow or reject incoming network traffic, depending on the sender and service.

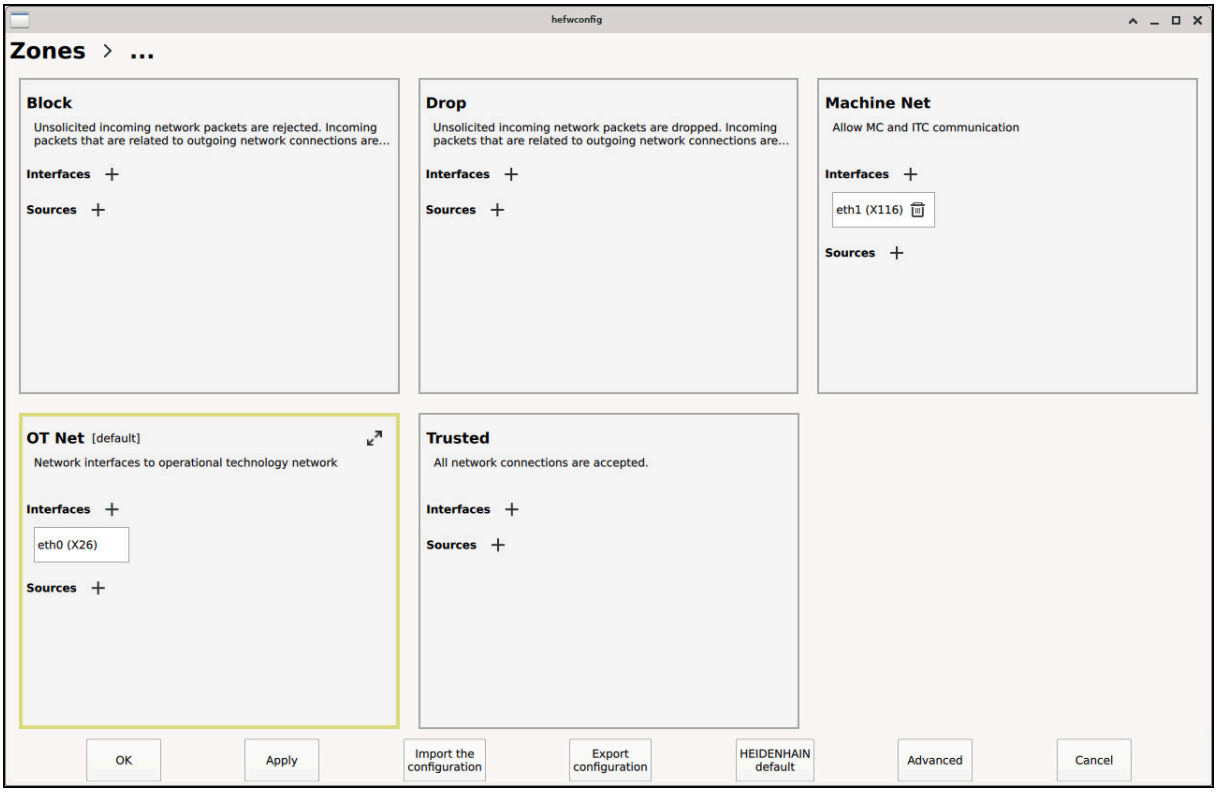
Related topics

- Existing network connection
  Further information: "Ethernet interface", Page 581
- SELinux security software
  Further information: "SELinux security software", Page 576
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 678

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Firewall



Overview of zones

The **hefwconfig** window always shows the **OT Net** zone after opening. If you select **Zones** in the breadcrumb navigation, the control opens an overview of the zones.

Five zones are displayed on the default overview screen. The overview shows which interfaces and sources have been added to the respective zones.

Each zone has its own default configuration.

Further information: "Zones", Page 616

You can edit the configuration of the zones.

Further information: "Settings of the zones", Page 617

For example, network specialists can make the following changes:

- Add and remove zones
- Rename zones
- Edit the zone description
- Edit the default target of the zone

If a network specialist has made changes to the firewall, the firewall of your machine might differ from the default configuration.

Icons and buttons

The **hefwconfig** window provides the following icons and buttons:

Icon or button	Meaning
	**Maximize** Open the selected zone
	**Reduce** Close the open zone and return to the overview screen
	**Add**
	**Clear**
	**Edit** Edit comprehensive rule
OK	Save the changes and close the window
Apply	Save the changes
Import the configuration	Import the configuration and overwrite the present configuration
Export configuration	Export the configuration of all zones
HEIDENHAIN default	Reset the settings to their default values
Advanced	Open the **Firewall Configuration** window Available only to network specialists
Cancel	Discard the changes that have not been saved and close the window

Default targets

Each zone has a default target. The default target defines how the firewall will handle incoming network connections. The firewall provides the following default targets:

Default target	Meaning
ACCEPT	Accept all incoming network connections This corresponds to the deactivation of the firewall.
DROP	Discard the incoming network connections You can add or remove exceptions.
REJECT	Reject the incoming network connections You can add or remove exceptions.

Further information: "Settings of the zones", Page 617

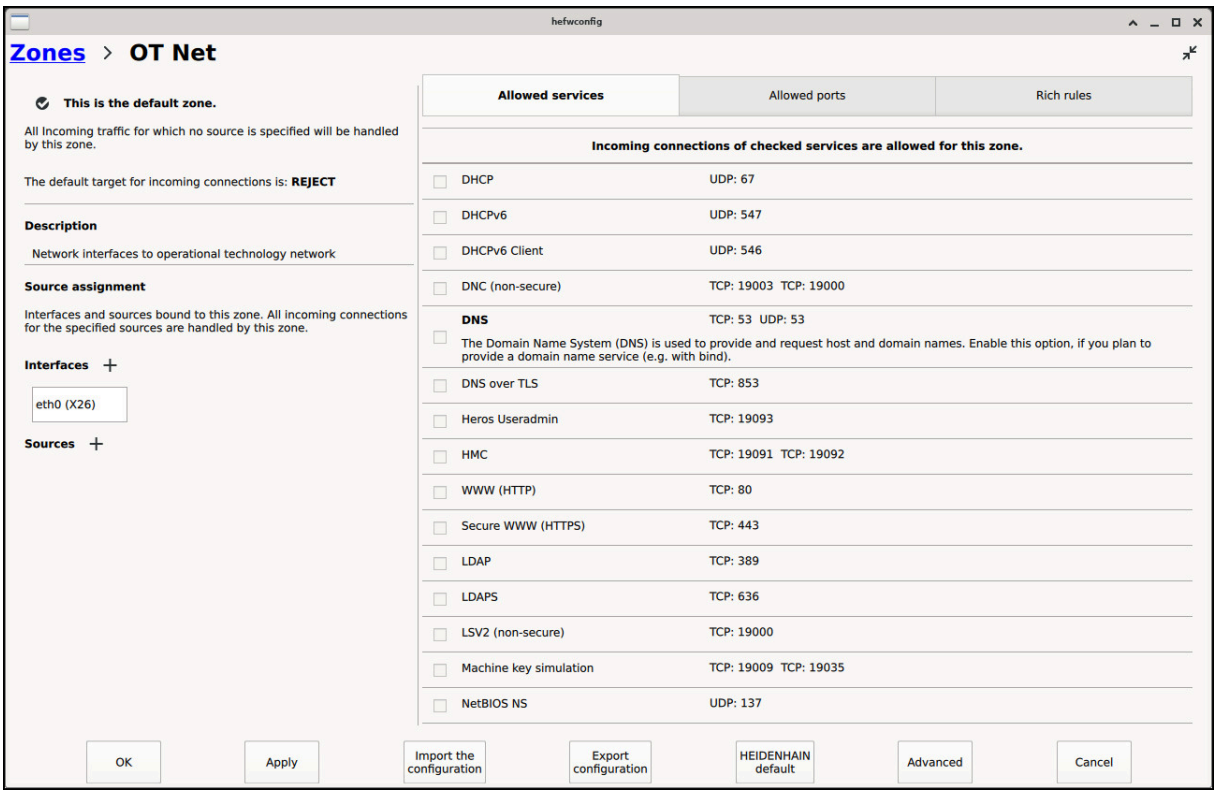
Zones

The following table shows the available zones and the default configuration:

Zone	Meaning
Block	Default target: **REJECT** This zone rejects all incoming connections.
Drop	Default target: **DROP** This zone discards all incoming connections.
Machine Net	Default target: **REJECT** with exceptions This zone accepts all the services needed for connections between the control and an additional ITC operating station (e.g., **VNC** or **DNS**). The **eth1** interface is assigned to this zone.
OT Net	This zone is the default zone. Default target: **REJECT** with exceptions This zone accepts the **SSH** service. The **eth0** interface is assigned to this zone.
Trusted	Default target: **ACCEPT** This zone accepts all incoming connections.

On programming stations, the **eth1** interface is assigned to the additional zone **Programmingstation Network** by default.

Settings of the zones



OT Net zone with description of the **DNS** service

When you open a zone, the control displays the following settings:

Setting	Meaning
Default zone	In this area, the control shows whether the zone is the default zone. If the zone is not the default zone, you can define this zone as the default zone by selecting the check box. The control automatically assigns all the unassigned interfaces and sources to the default zone.
Source assignment	In this area, the control shows the interfaces and sources assigned to this zone You can add or delete interfaces and sources.
Allowed services	On the **Allowed services** tab, the control displays all available services and the related ports. Use the check boxes to allow or reject services. If the check box is selected, the service is allowed. When you select a service, the control displays the appropriate description. HEIDENHAIN recommends that you add or delete exceptions only in the **OT Net** zone.
Allowed ports	On the **Allowed ports** tab, you can allow the TCP or UDP protocol. When you select the **Add** button, the control displays a window. Select TCP or UDP and define the port or the range of ports.

Setting	Meaning
Rich rules	On the **Rich rules** tab, you can define the exceptions for sources, services, and ports in more detail. When you create a comprehensive rule, the control provides the following selection options: ■ **Action** ■ **Accept** Accept the selected element ■ **Reject** Reject the selected element ■ **Drop** Discard the selected element ■ **Source** IP address or MAC address You can also define a rule using the element **Service**, **TCP**, or **UDP** without specifying a source. ■ **Element** ■ **All** You must specify a source. The selected action applies to all services and ports. ■ **Service** The control provides a selection menu containing all available services. ■ **TCP** The control provides an input field for the port or the range of ports. ■ **UDP** The control provides an input field for the port or the range of ports.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
- You must save all changes by using the **Apply** button; the control will discard the changes that have not been saved.
- You can also open a zone by double-tapping or double-clicking the zone.
- You can assign the interfaces or sources to different zones. A zone will be active once an interface or a source has been assigned to it.
- You can also add or delete interfaces and sources on the overview screen of the zones.
- If you delete an interface or source from a zone, the control will always assign this interface or source to the default zone. You cannot delete any interfaces or sources from the default zone.

24.21 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings
  Further information: "Firewall", Page 614
- Network settings
  Further information: "Network configuration with Advanced Network Configuration", Page 681

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **Portscan**

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine-manufacturer-specific functions: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

Start the portscan manually by selecting the **Start** button in the **Manual Execution** area. In the **Automatic Execution** area, you can use the **Automatic update on** function to specify that the control will perform the portscan automatically in the selected interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists may be open. The control shows a message window if a port is not listed.

24.22 Backup and restore

Application

The **NC/PLC Backup** and **NC/PLC Restore** functions allow you to back up and restore individual folders or the complete **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive
  Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ▶ Diagnostics/Maintenance ▶ NC/PLC Backup

Settings ▶ Diagnostics/Maintenance ▶ NC/PLC Restore

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double-tap or double-click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: Programming and Testing User's Manual

Within the backup function you can chose between the following types of backups:

- **Back up the “TNC:” partition**
  Back-up all data on the **TNC:** drive
- **Back up the directory tree**
  Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**
  Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**
  Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **Continue** and **Back** buttons.

24.22.1 Backing up data

To back-up the data of the **TNC:** drive:

- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the “TNC:” partition** window.
- Specify the type of backup
- Select **Continue**
- If necessary, pause the control with **Stop NC software**
- Select any predefined exclusion rules or ones you have defined yourself
- Select **Continue**
- > The control generates a list of files for backing up.
- Check list
- Deselect files if necessary

  Further information: "Selecting or deselecting multiple files", Page 622
- Select **Continue**
- Enter the name of the backup file
- Select the storage path
- Select **Continue**
- > The control generates the backup file.
- Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

24.22.2 Restoring data

NOTICE

Caution: Data may be lost!

When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted.

- Before starting the data restore process, make a backup of the existing data

To restore data:

- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Double-tap or double-click **NC/PLC Restore**
> The control opens the **Restore data - %1** window.
- Select the archive to be restored
- Select **Continue**
> The control generates a list of files for restoring.
- Check list
- Deselect files if necessary
  Further information: "Selecting or deselecting multiple files", Page 622
- Select **Continue**
- If necessary, pause the control with **Stop NC software**
- Select **Extract archive**
> The control restores the files.
- Confirm with **OK**
> The control restarts the NC software.

24.22.3 Selecting or deselecting multiple files

To select or deselect multiple files at a time:

- Select the first of the range of desired files.
- Select **Multiple selection**
> The control activates multiple selection.
- Select the last of the range of desired files
> The control highlights all files starting from the first selected file.
- Select the check box for selecting or deselecting
> The control selects or deselects all the highlighted files.
- Select **Multiple selection**
> The control deactivates multiple selection.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

24.23 TNCdiag

Application

TNCdiag displays status and diagnostic information of HEIDENHAIN components.

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **TNCdiag**

Only use **TNCdiag** after consultation with your machine manufacturer, unless you want to use it for setting up wireless handwheels.

Further information: "Setting up a wireless handwheel ", Page 534

For general information, please refer to the **TNCdiag** documentation.

24.24 Update the documentation

Application

The **Update the documentation** function can be used, for example, to install or update the integrated **TNCguide** product aid.

Related topics

- Integrated product aid **TNCguide**
  Further information: "User's Manual as integrated product aid: TNCguide", Page 53
- Product aid on the HEIDENHAIN website
  TNCguide

Description of function

To navigate to this function:

Settings ▶ **Diagnostics/Maintenance** ▶ **Update the documentation**

The file manager is located in the **Update the documentation** area. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 625

The control shows all available documents in the **Help** application.

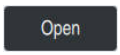
In the **Update the documentation** area, you can install all HEIDENHAIN-specific documents (e.g., NC error messages).

24.24.1 Transferring TNCguide

To find and transfer the desired **TNCguide** version:

- Select the link to the HEIDENHAIN website
  https://content.heidenhain.de/doku/tnc_guide/html/de/index.html
- Select **TNC controls**
- Select **TNC7 Series**
- Select the NC software number
- Navigate to the **product aid (HTML)**
- Select **TNCguide** in the desired language
- Select the path for saving the file
- Select **Save**
- > The download starts.
- Transfer the downloaded file to the control

- Select the **Home** operating mode
- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Select **Update the documentation**
- > The control opens the **Update the documentation** area.
- Select the desired file with the extension ***.tncdoc**



- Select **Open**
- > The control reports in a window whether installation was successful or failed.
- Select the **Help** application

- Select **Home**
- > The control displays all available documentation.

24.25 Machine parameters

Application

You can configure the behavior of the control with machine parameters. For this purpose, the control provides the **MPs for Users** and **MPs for setters** applications. You can open the **MPs for Users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

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

Requirements

- Code number 123
  Further information: "Code numbers", Page 561
- The contents of the **MPs for setters** application have been defined by the machine manufacturer

Description of function

To navigate to this function:

Settings ▶ **Machine Parameters** ▶ **MPs for setters**

In the **Machine Parameters** group the control shows only those menu items that you can choose with the current access rights.

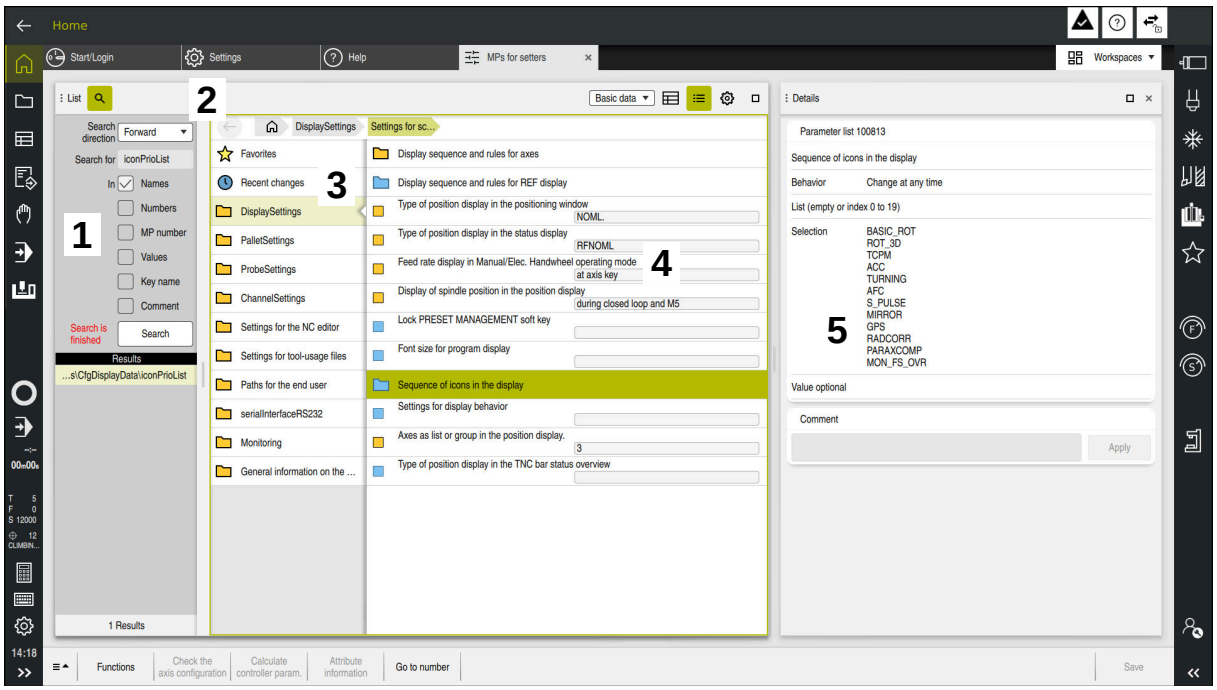
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **Details**
- **Document**
- **List**

You cannot close the **List** workspace.

The configuration editor areas



The **MPs for setters** application with a machine parameter selected

The configuration editor shows the following areas:

1 The **Search** column

You can search forward or backward with the following characteristics:

- Name

  This is the language-neutral name used for machine parameters in the User's Manual.
- Number

  This is the unique number used for machine parameters in the User's Manual.
- MP number of the iTNC 530
- Value
- Key name

  Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)
- Comment

The control displays the results.

2 Title bar of the **List** workspace

The title bar of the **List** workspace includes the following functions:

- Open or close the **Search** column
- Filter contents using a selection menu
- Toggle between structure and table views

  Further information: "The contents displayed in table view", Page 630
- Open or close the **Details** workspace

  Further information: "The Details workspace", Page 631
- Open or close the **Configuration** window

  Further information: "The Configuration window", Page 630

3 Navigation column

The control provides the following options for navigation:

- Navigation path
- Favorites
- 21 most recent changes
- Structure of the machine parameters

4 Content column

In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.

5 The **Details** workspace

The control displays information on the selected machine parameter or the most recent change you made.

Further information: "The Details workspace", Page 631

Icons, buttons and shortcuts

The configuration editor provides the following icons, buttons, and shortcuts:

Icon, button, or shortcut	Meaning
	Activate or deactivate the **table view** The control toggles between structure and table views. **Further information:** "The contents displayed in table view", Page 630
	Activate or deactivate **Change column width** You can adjust the width of the currently selected column. Only if table view is active
	Open or close the **Details** workspace **Further information:** "The Details workspace", Page 631
	Open or close the **Configuration** window **Further information:** "The Configuration window", Page 630
	Select **Recent changes**
	Object exists - Data object - Directory - Parameter list
	Object empty
	Machine parameter exists
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter not readable and not editable

Icon, button, or shortcut	Meaning
	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** Programming and Testing User's Manual
CTRL + N	Create a new object in a list of data objects or parameters
Check the axis configuration	Only for the machine manufacturer
Calculate controller param.	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Go to number	The control opens the **Enter number, then select from list** window and suggests the number of the currently selected object. You can enter the number of a machine parameter and navigate directly to this parameter.
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

The contents displayed in table view

The **List** workspace in table view

The table view of the **List** workspace displays the following information:

1 Group name (key)
The key is displayed in the header of the table. If no key is available, the header is empty.

2 Object name (entity)
The entity is displayed left-aligned in the first column. The name of the entity starts with **Cfg**.

3 Name (attribute) of the machine parameter
The attribute is displayed right-aligned in the first column.

4 Index of the machine parameter
If indexes are available, the index numbers are enclosed within square brackets.

The table view allows you to compare the configurations of different keys, for example.

The Configuration window

The control provides the following toggle switches in the **Configuration** window:

- **Show MP descriptive texts**
  If the toggle switch is active, the control displays a description of the machine parameter in the active conversational language.
  If the toggle switch is not active, the control displays the language-neutral name of the machine parameter.
- **Display in tree view**
  If the toggle switch is active, the control displays the machine parameters in a tree view.
  If the toggle switch is not active, the control displays the machine parameters in the structure view.

The Details workspace

If you select contents from the favorites or the structure, the control will display information in the **Details** workspace, such as:

- Type of object, such as data object list or parameter
- Descriptive text of machine parameter
- Permitted or required input
- Prerequisite for the change (e.g., program run blocked)
- Number of the machine parameter on the iTNC 530
- Machine parameter optional

  This information is included if a machine parameter can be enabled optionally.

If you select contents from the most recent changes you made, the control will display the following information in the **Details** workspace:

- Sequential number of the last change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Permitted or required input

Notes

- The **CFGREAD** function allows you to read the values from machine parameters. Use the table view to determine the parameters required for **CFGREAD**.

  Further information: Programming and Testing User's Manual
- The machine manufacturer defines which machine parameters are saved user-specifically by the control when user administration is active. These machine parameters can be changed at any time without, for example, having to restart the control.

  Further information: "User administration", Page 635
- The machine manufacturer offers further applications for machine parameters.
- If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

24.26 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is customized.

Related topics

- Workspaces
  Further information: "Workspaces", Page 85
- Control interface
  Further information: "Areas of the control's user interface", Page 82

Description of function

To navigate to this function:

Settings ▶ Configurations ▶ Configurations

A configuration contains all adaptations to the control's user interface that do not influence the control's actual functions.

- Settings for the TNC bar
- Arrangement of workspaces
- Font size
- Favorites

The **Configurations** area contains the following functions:

Function	Meaning
Active Configuration	Activate a configuration from a selection menu **Further information:** "The Desktop menu workspace", Page 100
Default configuration	Use the **Reset** button to apply the settings of the **OEM configuration** to the active configuration.
Save as OEM Configuration	The machine manufacturer can use the **Save** button to overwrite the **OEM configuration**.
Save current settings	With the **Save** button, you can save the current version of the active configuration.
Restore last configuration	With the **Reset** button, you can discard any customizations and revert to the saved version of the active configuration.

The control displays the following information about all available configurations in a table:

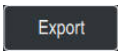
Column	Meaning
Configuration Name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 633
Edit	This column contains two buttons, for renaming and deleting the configuration.

Press the **Add** button to create a new configuration.

24.26.1 Exporting and importing configurations

To export configurations:

- Select the **Settings** application
- Select **Configurations**
- > The control opens the **Configurations** area.
- Activate the **Exportable** toggle switch for the desired configuration, if necessary



- Select **Export**
- > The control opens the **Save as** window.
- Select the target file
- Enter a file name

- Select **Create**
- > The control saves the configuration file.

To import configurations:

- Select **Import**
- > The control opens the **Import configurations** window.
- Select file

- Select **Import**
- > If importing a configuration would overwrite a file with the same name, the control displays a prompt.
- Select the procedure:
  - **Overwrite**: The control overwrites the original configuration.
  - **Keep**: The control does not import the configuration.
  - **Cancel**: The control cancels the import process.

Notes

- Delete only inactive configurations. If you delete an active configuration, the control first actives a default configuration. This can lead to delays.
- The **Overwrite** function permanently replaces existing configurations.

25

User administration

25.1 Fundamentals

Application

User administration enables you to create and administrate different users with different access rights to various functions of the control. You can assign roles to the various users that reflect their respective tasks, such as machine operator or setup technician.

User administration is inactive in the control's factory default setting. This status is called **Legacy-Mode**.

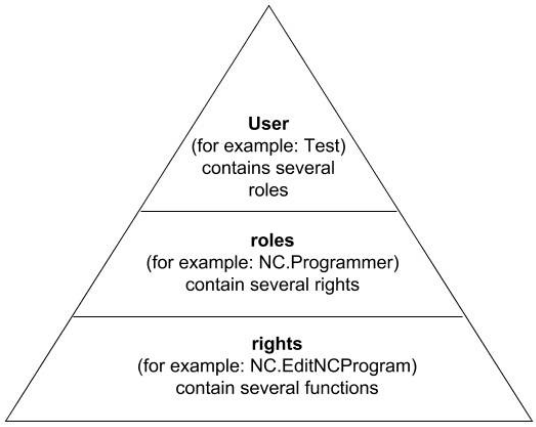
Description of function

User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:

- Application security
- Network security
- Platform security

The user administration differentiates between the following terms:

- User
  Further information: "Users", Page 636
- Roles
  Further information: "Roles", Page 638
- Rights
  Further information: "Rights", Page 638



Users

The user administration offers the following types of users:

- Function users pre-defined by HEIDENHAIN
- Function users pre-defined by the machine manufacturer
- Self-defined users

Depending on the task assigned, you can use one of the pre-defined function users or you have to create a new user.

Further information: "Creating a new user", Page 642

If you deactivate user administration, the control saves all configured users. Thus they will be available again when user administration is reactivated.

If you want to delete the configured users upon deactivation, you need to set this explicitly when deactivating user administration.

Further information: "Deactivating user administration", Page 643

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.

- **sys**

  The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.

- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.

- **oem**

  The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer can define up to 32 function users, such as for machine maintenance or for setting up and operating external systems.

Function users defined by the machine manufacturer can also be used as a substitute for code numbers. You can use the function users' passwords to enable their additional rights temporarily.

Further information: "The Active user window", Page 645

The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.

Roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

Further information: "List of roles", Page 687

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

User administration offers roles for the following tasks:

- **Operating system roles:** access to functions of the operating system and interfaces
- **NC operator roles:** access to functions for programming, setting up and running NC programs
- **Machine tool builder (PLC) roles:** access to functions for configuring and checking the control

Every user should have at least one role from the operating system area and at least one role from the programming area.

HEIDENHAIN recommends permitting more than one person to access an account with the HEROS.Admin role. This ensures that necessary changes to user administration can also be made in the administrator's absence.

Local or remote registration

You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.

Further information: "SSH-secured DNC connection", Page 658

If a role is only enabled for local login, "Local." is added to the role name (e.g., Local.HEROS.Admin instead of HEROS.Admin).

If a role is only enabled for remote login, "Remote." is added to the role name (e.g., Remote.HEROS.Admin instead of HEROS.Admin).

You can therefore also make the rights of a user dependent on the access used to operate the control.

Rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.

Rights gather various functions of the control (e.g., editing the tool table).

User administration offers rights for the following tasks:

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.

Ensure that every user is assigned all access rights he needs. The access rights result from the tasks a user performs on the control.

The access rights of HEIDENHAIN function users are already pre-defined in the control's factory default setting.

Further information: "List of rights", Page 690

Password settings

If you use an LDAP database, users with the HEROS.Admin role can define password requirements. For this, the control provides the **Password settings** tab.

Further information: "Saving user data", Page 646

The following parameters are available:

Password lifetime

- **Validity period of password:**
  Here, you can indicate how long the password can be used.
- **Warning before expiration:**
  From the defined time, a warning will be issued that the password will soon expire.

Password quality

- **Minimum password length:**
  Here, you can indicate the minimum password length.
- **Minimal number of character classes (upper/lower, digits, special):**
  Here, you can indicate the minimum number of different character classes required in the password.
- **Maximum number of repeated characters:**
  Here, you can indicate the maximum number of identical successive characters in the password.
- **Maximum length of character sequences:**
  Here, you can indicate the maximum length of the character sequences to be used in the password (e.g., 123).
- **Dictionary check (number of matching characters):**
  Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.
- **Minimum number of characters changed compared to previous password:**
  Here, you can specify how many characters in the new password must be different from the previous one.

You define the values for each parameter on a scale.

For reasons of security, passwords should comply with the following criteria:

- Eight characters minimum
- Letters, numbers, and special characters
- Avoid using whole words or a sequence of characters (e.g., Anna or 123)

If you want to use special characters, pay attention to the keyboard layout. HEROS assumes a US keyboard, the NC software assumes a HEIDENHAIN keyboard. External keyboards can be freely configured.

Additional directories

HOME: drive

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by the respectively logged-in users as well as users with the HEROS.Admin role.

public directory

Upon the first activation of user administration, the **public** directory below the **TNC:** drive will be connected.

The **public** directory can be accessed by any user.

In the **public** directory you can, for example, make files available to other users.

25.1.1 Configuring user administration

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

1 Open the **User administration** window
2 Activating user administration
3 Defining the password for the **useradmin** function user
4 Setting up a database
5 Creating a new user

- You can exit the **User administration** window after each configuration step.
- If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Open the User administration window

To open the **User administration** window:

- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **CurrentUser**
- > The control opens the **User administration** window in the **Settings** tab.

Further information: "The User administration window", Page 644

Activating user administration

To activate user administration:

- Select **User administration active**
- > The control shows the message **Password for user 'useradmin' missing**.
- Retain or reactivate the active status of the **Anonymize users in log data** function

- The purpose of the **Anonymize users in log data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.
- If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Defining the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 636

To define a password for the **useradmin** function user:

- Select **Password for useradmin**
- > The control opens the **Password for user 'useradmin'** pop-up window.
- Enter the password for the **useradmin** function user

Please observe the recommendations for passwords.

Further information: "Password settings", Page 639

- Repeat the password
- Select **Set new password**
- > The control shows the message **Settings and password for 'useradmin' were changed**.

Setting up a database

To set up a database:

- Select the database for saving your user data (e.g., **Local LDAP database**)
- Select **Configuration**
> The control opens a window for configuring the corresponding database.
- Follow the instructions from the control in the window
- Select **APPLY**

> The following options are available for saving your user data:
> - **Local LDAP database**
> - **LDAP on remote computer**
> - **Connection to Windows domain**
>
> Parallel operation of Windows users and users from an LDAP database is possible.
>
> **Further information:** "Saving user data", Page 646

Creating a new user

To create a new user:

- Select the **User administration** tab
- Select **Create new user**
> The control adds a new user to the **User list**.
- Change the name as needed
- Edit a password as needed
- Define a profile image as needed
- Enter a description as needed
- Select **Add role**
> The control opens the **Add role** window.
- Select a role
- Select **Add**

> You can also add roles using the **Add external login** and **Add local login** buttons.
>
> **Further information:** "Roles", Page 638

- Select **Close**
> The control closes the **Add role** window.
- Select **OK**
- Select **APPLY**
> The control adopts the changes.
- Select **END**
> The control opens the **System reboot required** window.
- Select **Yes**
> The control restarts.

> The user must change the password when logging in for the first time.

25.1.2 Deactivating user administration

User administration can be deactivated only by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 636

To deactivate user administration:

- ▶ Log in as a function user
- ▶ Open the **User administration** window
- ▶ Select **User administration inactive**
- ▶ If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
- ▶ Select **APPLY**
- ▶ Select **END**
- > The control opens the **System reboot required** window.
- ▶ Select **Yes**
- > The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in log data** function, the system will show personalized user data in all control log files.
If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case, it is your responsibility to ensure that all required data protection provisions have been made at your company.

- ▶ Retain or reactivate the active status of the **Anonymize users in log data** function

- Some user administration areas are configured by the machine manufacturer. Refer to your machine manual.
- HEIDENHAIN recommends activating user administration as part of an IT safety concept.
- If both user administration and a screensaver are active, then the current user's password must be entered to unlock the screen.

  Further information: "HEROS menu", Page 663
- If you used **Remote Desktop Manager** to establish private connections before user administration was activated, these connections are no longer available after the activation of user administration. Save your private connections before activating user administration.

  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608

25.2 The User administration window

Application

In the **User administration** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- The **Active user** window
  Further information: "The Active user window", Page 645

Requirement

- If user administration is active, the HEROS.Admin role
  Further information: "List of roles", Page 687

Description of function

To navigate to this function:

Settings ▶ **Operating System** ▶ **UserAdmin**

The **User administration** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration **Further information:** "Configuring user administration", Page 640
User administration	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 642
Password settings	Define password requirements **Further information:** "Password settings", Page 639
User-defined roles	Roles created for a Windows domain **Further information:** "Connection to Windows domain", Page 648

25.3 The Active user window

Application

In the **Active user** window, the control displays information about the logged on user, such as assigned rights. You can also manage other user settings, such as keys for SSH-secured DNC connections or smartcards for logon, and change the password.

Related topics

- SSH-secured DNC connections
  Further information: "SSH-secured DNC connection", Page 658
- Logon with smartcards
  Further information: "Logon with smartcards", Page 654
- Available roles and rights
  Further information: "User administration roles and rights", Page 687

Description of function

To navigate to this function:

Settings ▶ **Operating System** ▶ **Current User**

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the **Added rights** tab, enable rights for another user or function user until the next logoff
Open user administration	Open the **User administration** window **Further information:** "The User administration window", Page 644
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 658 **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590
Create token	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 654
Delete token	
Close	Close the **Active user** window

On the **Change password** tab you can check your password against the current requirements or set a new password.

Further information: "Password settings", Page 639

Note

In legacy mode, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect.

Further information: "Users", Page 636

25.4 Saving user data

25.4.1 Overview

The following options are available for saving your user data:

- **Local LDAP database**
  Further information: "Local LDAP database", Page 646
- **LDAP on remote computer**
  Further information: "LDAP database on a remote computer", Page 647
- **Connection to Windows domain**
  Further information: "Connection to Windows domain", Page 648

> Parallel operation of Windows users and users from an LDAP database is possible.

25.4.2 Local LDAP database

Application

With the **Local LDAP database** setting the control saves the user data locally. That way you can activate user administration even on machines without a network connection.

Related topics

- Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 647
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 648

Requirements

- User administration is active
  Further information: "Activating user administration", Page 641
- **useradmin** user is logged on
  Further information: "Users", Page 636

Description of function

A local LDAP database offers the following options:

- Using user administration on one single control
- Setting up a central LDAP server for more than one control
- Exporting an LDAP server configuration file if the exported database is to be used by more than one control

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- Open the **User administration** window
- Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- Select **Local LDAP database**
- Select **Configuration**
- > The control opens the **Configure local LDAP database** window.
- Enter the name of the **LDAP domain**
- Enter the password
- Repeat the password
- Select **OK**
- > The control closes the **Configure local LDAP database** window.

Notes

- Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.

  Passwords must not be trivial and must be known only to the administrators.
- If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

25.4.3 LDAP database on a remote computer

Application

With the **LDAP on remote computer** function you can transmit the configuration of a local LDAP database between controls and computers. That way you can use the same users on multiple controls.

Related topics

- Configuring an LDAP database on a control

  Further information: "Local LDAP database", Page 646
- Connecting a Windows domain with user administration

  Further information: "Connection to Windows domain", Page 648

Requirements

- User administration is active

  Further information: "Activating user administration", Page 641
- **useradmin** user is logged on

  Further information: "Users", Page 636
- LDAP database has been set up in the company network
- Server configuration file of an existing LDAP database is stored on the control or a PC in the network

  If the configuration file is stored on a PC, the PC must be running and accessible through the network.

  Further information: "Providing a server configuration file", Page 648

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:

- Open the **User administration** window
- Select **LDAP user database**
> The control enables the dimmed area for editing the LDAP user database.
- Select **Local LDAP database**
- Select **Export server configuration**
> The control opens the **Export LDAP configuration file window.**
- Enter the name for the server configuration file into the name field
- Save the file to the desired folder
> The control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:

- Open the **User administration** window
- Select **LDAP user database**
> The control enables the dimmed area for editing the LDAP user database.
- Select **LDAP on remote computer**
- Select **Import server configuration**
> The control opens the **Import LDAP configuration file window.**
- Select the existing configuration file
- Select **FILE**
- Select **APPLY**
> The control imports the configuration file.

25.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function, you can connect the data of a domain controller with the control's user administration.

Ask your IT administrator to configure the connection to the Windows domain.

Related topics

- Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 646
- Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 647

Requirements

- User administration is active
  Further information: "Activating user administration", Page 641
- **useradmin** user is logged on
  Further information: "Users", Page 636
- Windows domain controller present in the network
- Domain controller accessible in the network
- Organizational unit for HEROS roles known
- Function user is defined in the organization
- User name and password of the function user are known

Description of function

Your IT administrator sets up a function user for connecting to the Windows domain.

Further information: "Joining a Windows domain", Page 652

Buttons

The **Connection to Windows domain** area provides the following buttons:

Button	Meaning
Configuration	The control opens the **Configure Windows domain with function user** window. **Further information:** "The Configure Windows domain with function user window", Page 650
Find domain	The control selects a Windows domain.
Export the Windows config.	Once you have connected the control to the Windows domain, you can export the configurations for other controls. **Further information:** "Exporting and importing a Windows configuration file", Page 652
Import the Windows config.	Using a present configuration, you can connect the control easily and quickly to the Windows domain. **Further information:** "Exporting and importing a Windows configuration file", Page 652
Check missing role definitions	The control checks whether all of the required roles have been created in the Windows domain.
Add role definition	If any roles required in the Windows domain are missing, you can add the missing roles. **Further information:** "Groups of the domain", Page 651

The Configure Windows domain with function user window

After the domain search, you can customize the Windows domain information or specify new information in the **Configure Windows domain with function user** window.

Your IT administrator will provide the required information.

The **Configure Windows domain with function user** window provides the following settings:

Setting	Meaning
Domain name:	Server name of the Windows domain Is populated by domain search
Key Distribution Center (KDC):	KDC address Is populated by domain search
Alternative admin server:	Deviating server name where the passwords are managed
Map SIDs to Unix UIDs	Map the Windows user SIDs (Security IDs) in Active Directory to the matching Unix UIDs on the control
Use LDAPs	Transfer data using secure LDAPs LDAPs encrypt user data and passwords. You can select a certificate or disable certificate validation.
Group for login authorization:	Define a special group of Windows users to whom you want to restrict the connection to this control
Organizational unit for HEROS roles:	Modify the organizational unit in which the HEROS role names are stored Specify the configuration of your domain.
Prefix for HEROS role names:	Change the prefix in order to manage users from different workshops, for example. Each prefix given to a HEROS role name can be changed (e.g., HEROS hall 1 and HEROS hall 2) Is populated by domain search
Separator for HEROS role names:	Modify the separator within the HEROS role names
Function user:	User name and password of the Active Directory function user
Organizational unit for function user:	Organizational unit of the function user
Advanced configuration of domain section	Only for IT administrators

The function user's user name must not contain blanks. The name and organizational unit form the complete path (Distinguished Name, DN) in the Active Directory.

Groups of the domain

If not all of the required roles have been created in the domain as groups, the control issues a warning.

If the control issues a warning, proceed in one of the two following ways:

- Use the **Add role definition** function to enter a role directly in the domain
- Use the **Export role definition** function to export the roles to an ***.ldif** file

There are the following ways to create groups corresponding to the different roles:

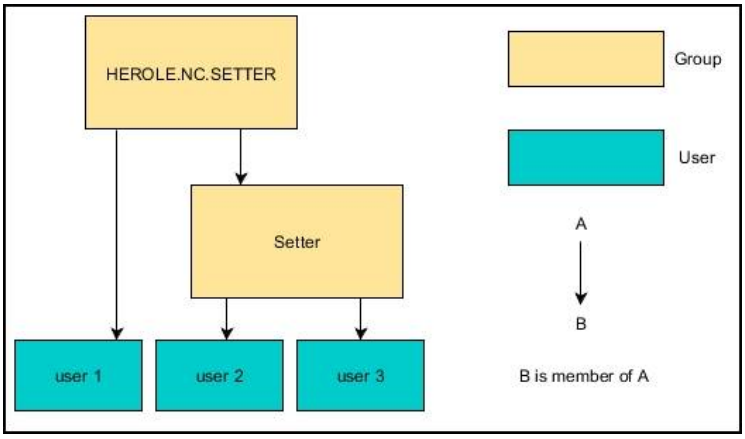
- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

The Windows administrator must add the users manually to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by below.

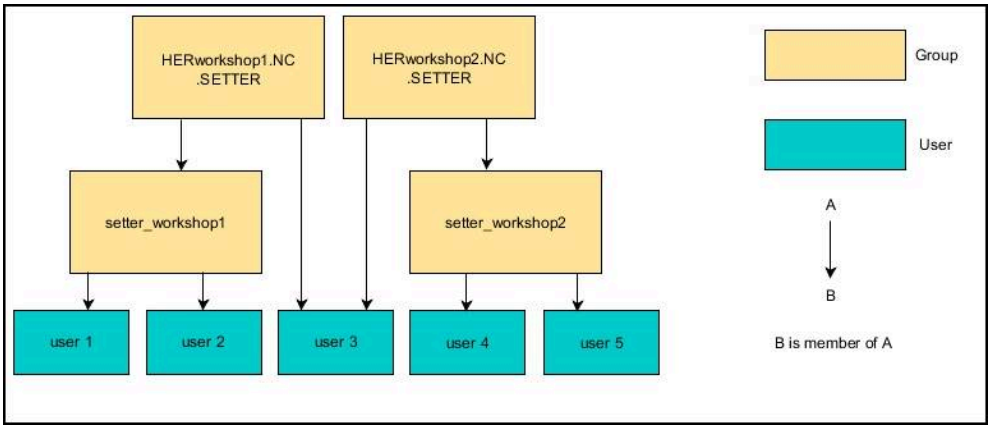
Example 1

The user is a direct or indirect member of the respective group:



Example 2

Users from various sectors (workshops) are members of groups with different prefixes:



Joining a Windows domain

To join a Windows domain:

- Open the **User administration** window
- Select **Connection to Windows domain**
- Select **Find domain**
- > The control selects a domain.
- Select **Configuration**
- Check the data for **Domain name:** and **Key Distribution Center (KDC):**
- Enter **Organizational unit for HEROS roles:**
- Enter the user name and password of the function user
- Press **OK**
- Select **APPLY**
- > The control connects to the Windows domain found.
- > The control checks whether all of the required roles have been created in the domain as groups.

Exporting and importing a Windows configuration file

If you have connected the control to the Windows domain, you can export the required configurations for other controls.

To export the Windows configuration file:

- Open the **User administration** window
- Select **Connect to Windows domain**
- Select **Export the Windows config.**
- > The control opens the **Export the Windows domain configuration** window.
- Select the directory for the file
- Enter the name for the file
- Select the **Export the function user's password?** check box, if required
- Select **Export**
- > The control saves the Windows configuration as a BIN file.

To import the Windows configuration file of another control:

- Open the **User administration** window
- Select **Connect to Windows domain**
- Select **Import the Windows config.**
- > The control opens the **Import the Windows domain configuration** window.
- Select the existing configuration file
- Select the **Import the function user's password?** check box, if required
- Select **Import**
- > The control adopts the configurations for the Windows domain.

25.5 Autologin in user administration

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

- User login
  Further information: "Logging on with user administration", Page 654
- Configuring user administration
  Further information: "Configuring user administration", Page 640

Requirements

- User administration has been configured
- The user for **Autologin** has been defined

Description of function

With the **Enable autologin** check box in the **User administration** window, you can define a user for autologin.

Further information: "The User administration window", Page 644

The control then automatically logs this user on and displays the user interface according to the defined rights.

For further authorizations, the control still requires an authentication to be entered.

Further information: "Window for requesting additional rights", Page 657

25.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

- Automatic user logon
  Further information: "Autologin in user administration", Page 653

Requirements

- User administration has been configured
- For logon with smartcards:
  - Euchner EKS card reader
  - Smartcard assigned to a user
    Further information: "Assigning a smartcard to a user", Page 655

Description of function

The control displays the Login dialog in the following cases:

- After the **User logout** function
- After the **Switch users** function
- After the **Lock display** function
- Immediately after control startup if user administration is active and **Autologin** is not enabled

When user administration is active, the control provides these functions in the **Start/Login** application and in the HEROS menu.

Further information: "HEROS menu", Page 663

The logon dialog gives you the following options:

- Users who logged in at least once
- **Other** user

Logon with smartcards

You can save a user's logon data on a smartcard and then log the user on with a card reader, without needing to enter a password. You can define whether a PIN is necessary for logon.

The card reader is attached over a USB port. You assign the smartcard to a reader as a token.

Further information: "Assigning a smartcard to a user", Page 655

The smartcard also has additional memory space, where the machine manufacturer can store his own user-specific data.

25.6.1 Logging on a user with password

To logon a user the first time:

- Select **Other** in the login dialog
- > The control enlarges the user icon you selected.
- Enter the user name
- Enter the user's password

> The control shows in the Login dialog whether CAPS LOCK is active.

- > The control opens a window with the message **Password expired. Change the password now.**
- Enter the current password
- Enter a new password
- Repeat the new password
- > The control uses the new user to log you in.
- > The control displays this user in the dialog during the next logon procedure.

25.6.2 Assigning a smartcard to a user

To assign a smartcard to a user:

- Insert a blank smartcard in the card reader
- Logon the desired smartcard user in user administration
- Select the **Settings** application
- Select **Operating System**
- Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- Select **Create token**
- > The control opens the **Write certificate to token** window.
- > The control displays the smartcard in the **Select token** area.
- Select the smartcard as the token to be written
- Select the **PIN protection?** check box, if required
- Enter user password (and PIN, if desired)
- Select **Start writing**
- > The control saves the user's logon data on the smartcard.

Notes

- You must restart the control in order for it to detect a card reader.
- You can overwrite smartcards that already contain information.
- If you change a user's password, you must reassign the smartcard.

25.7 Window for requesting additional rights

Application

If you do not have the rights required for a specific **HEROS menu** item, the control opens the window for requesting additional rights.

In this window, you can temporarily obtain more rights by adding another user's rights.

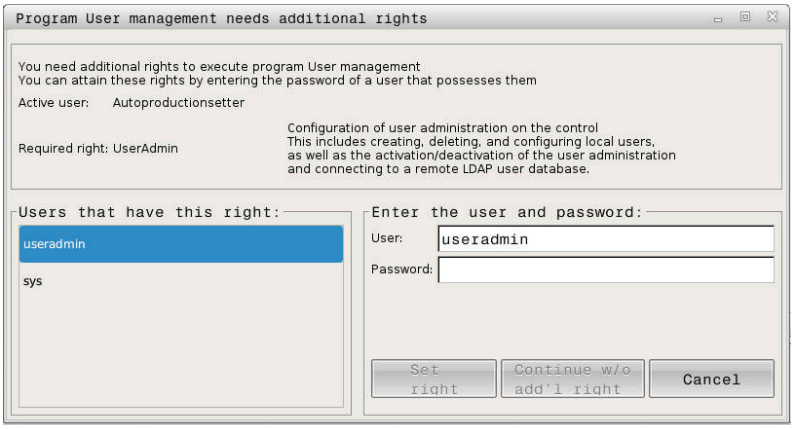
Related topics

- Temporarily granting additional rights in the **Active user** window
  Further information: "The Active user window", Page 645

Description of function

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

You must enter the password in order to enable user rights.



Window for requesting additional rights

To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Notes

- If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
- You can't use this window to change user administration settings. The user with the HEROS.Admin role must be logged on in order to do so.

25.8 SSH-secured DNC connection

Application

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

For DNC connections using the RPC or LSV2 protocol, the connection is routed through an SSH tunnel. This method assigns the remote user to a user set up on the control, granting the remote user this user's rights.

Related topics

- Forbidding non-secure connections
  Further information: "Firewall", Page 614
- Roles for remote logon
  Further information: "Roles", Page 638

Requirements

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

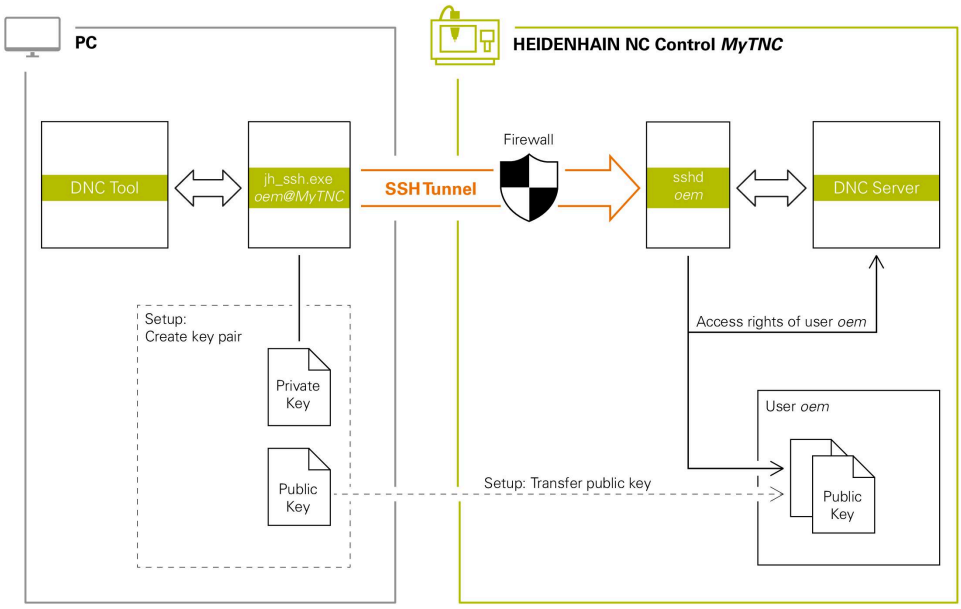
Description of function

Concept of transmission through an SSH tunnel

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to protect the connection. This key pair is generated on the client. The key pair consists of a private key and a public key. The private key remains with the client. During setup, the public key is transferred to the server and assigned to a certain user.

The client tries to connect to the server using the pre-defined user name. The server can use the public key to verify that the requester of the connection holds the associated private key. If yes, the server accepts the SSH connection and assigns it to the user that has been used for the login. Communication can then be "tunneled" through this SSH connection.



Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, establishing, and managing secure connections through an SSH tunnel.

When the connection is set up, the required key pair is generated in TNCremo and the public key is transferred to the control.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.

In order to expand the connection configuration using the associated **CreateConnections** tool, you need to update to **HEIDENHAIN DNC v1.7.1**. A modification of the application source code is not required.

25.8.1 Setting up SSH-secured DNC connections

To set up an SSH-secured DNC connection for the logged-on user:

- Select the **Settings** application
- Select **Network/Remote Access**
- Select **DNC**
- Activate the **Setup permitted** toggle switch
- Use **TNCremo** to set up the secure connection (TCP secure).

> For details, refer to the integrated help system of TNCremo.

> TNCremo transmits the public key to the control.

> In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

- Deactivate the **Setup permitted** toggle switch

25.8.2 Removing a secure connection

If you delete a private key from the control, that user no longer has the possibility of a secure connection.

To delete a key:

- Select the **Settings** application
- Select **Operating System**
- Double-tap or double-click **Current User**

> The control opens the **Active user** window.

- Select **Certificate and keys**
- Select the key to be deleted
- Select **Delete SSH key**

> The control deletes the selected key.

Notes

- The encryption used with the SSH tunnel protects the communication from attackers.
- For OPC UA connections, a stored user certificate is used for authentication.
  Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transfer a public key to the control by using a USB device or network drive.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

26

HEROS operating system

26.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 features the version HEROS 5.

26.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen.

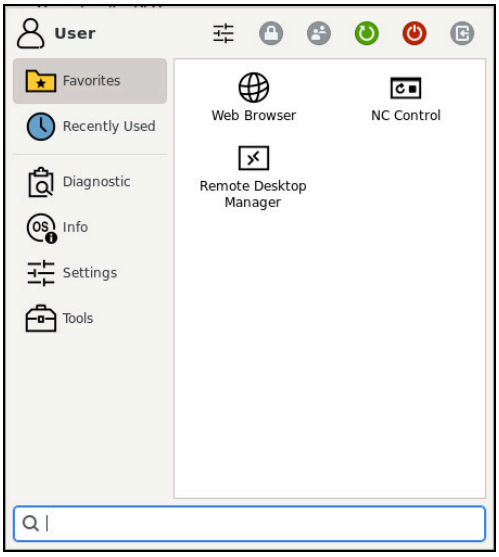
Related topics

- Opening HEROS functions through the **Settings** application
  Further information: "The Settings application", Page 557

Description of function

You open the HEROS menu with the green DIADUR icon in the task bar or with the **DIADUR** key.

Further information: "Taskbar", Page 667



Standard view of the HEROS menu

The HEROS menu contains the following functions:

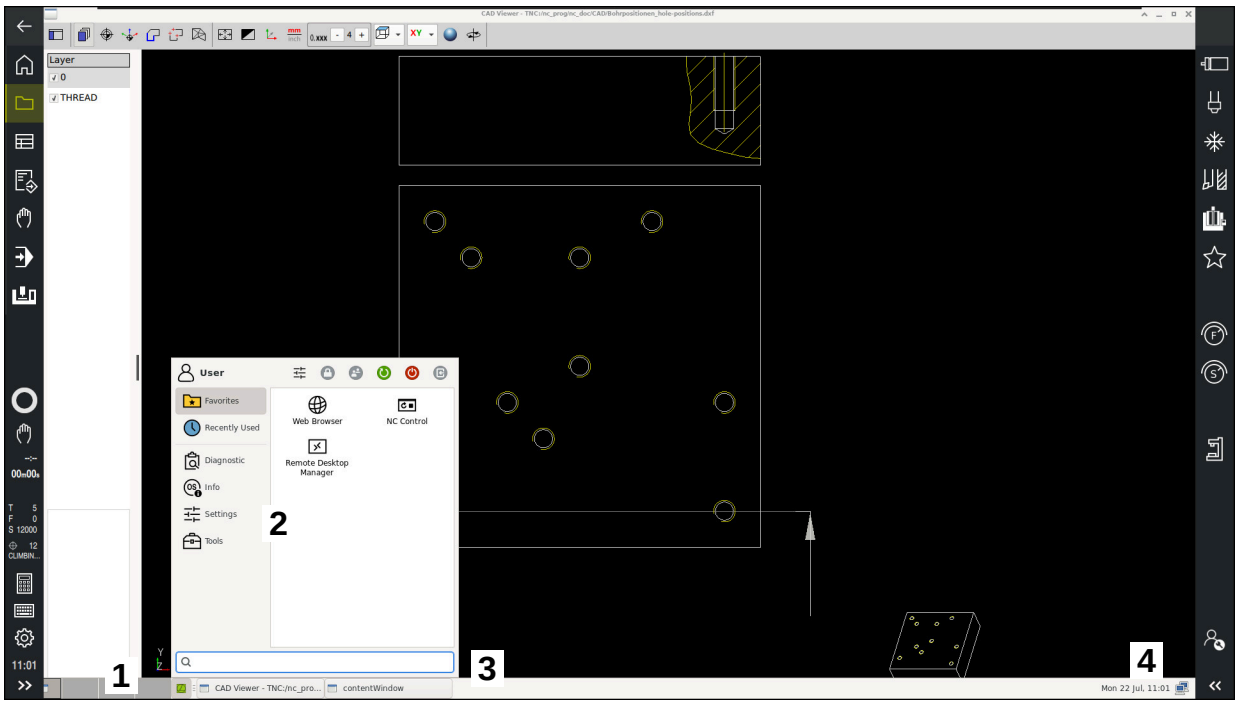
Area	Function
Header	■ User name **Further information:** "The Active user window", Page 645 ■ User-specific settings ■ Lock display Only if user administration is active ■ Switch users Only if user administration is active ■ Restart ■ Shut down ■ Log out Only if user administration is active **Further information:** "User administration", Page 635

Area	Function
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HeLogging**: Define settings for internal diagnostic files ■ **ITC VNC**: Display the screen contents of the additional operating station (ITC) ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 619 ■ **Portscan OEM**: Available only to authorized specialists ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically **Further information:** "TNCdiag", Page 623 ■ **TNCscope**: Available only to authorized specialists

Area	Function
Settings	■ **Adjust screen brightness**: Adjust screen brightness ■ **Screensaver**: Screensaver ■ **Current User** **Further information:** "The Active user window", Page 645 ■ **Date/Time** **Further information:** "The Adjust system time window", Page 573 ■ **Firewall** **Further information:** "Firewall", Page 614 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 574 ■ **Network** **Further information:** "Ethernet interface", Page 581 ■ **OEM Function Users** **Further information:** "Users", Page 636 ■ **OPC UA NC Server Connection Assistant** **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 596 ■ **OPC UA NC Server License** **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 597 ■ **PKI Admin**: Manage certificates for the control, such as for **OPC UA NC Server** **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 590 ■ **Printer** **Further information:** "Printers", Page 601 ■ **Reset Touchscreen Calibration** ■ **Screenshot Config** In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering). ■ **SELinux** **Further information:** "SELinux security software", Page 576 ■ **Shares** **Further information:** "Network drives on the control", Page 577 ■ **Touchscreen Calibration** ■ **Touchscreen Configuration** ■ **UserAdmin** **Further information:** "The User administration window", Page 644 ■ **VNC** **Further information:** "The VNC menu item", Page 605 ■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 668
Info	■ **About HeROS**: Open information about the operating system of the control ■ **About Xfce**: Open information on the Window manager

Area	Function
Tools	■ **Switch-off**: Shut-down or restart ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Document Viewer**: Display and print files (e.g., PDF files) ■ **Geeqie**: Open, manage, and print graphics ■ **Gnumeric**: Open, edit, and print tables ■ **Hostkey**: Show a unique ASCII image to identify the control ■ **IDS Camera Manager**: Manage cameras connected to the control ■ **keypad horizontal**: Open virtual keyboard ■ **keypad vertical**: Open virtual keyboard ■ **Leafpad**: Open and edit text files ■ **Mozilla Firefox**: Start the browser ■ **NC Control**: Start or stop the NC software independently of the operating system ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 620 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 620 ■ **Real VNC Viewer**: Connect to a remote device via a VNC server Available only to network specialists ■ **Remote Desktop Manager** **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 608 ■ **Ristretto Image Viewer**: Open graphics ■ **Secure Remote Access** **Further information:** "Secure Remote Access", Page 675 ■ **Combine fixtures** **Further information:** "Combining fixtures in the New Fixture window", Page 271 ■ **Touchscreen Cleaning** ■ **Web Browser**: Start the browser ■ **Xarchiver**: Extract or compress directories
Search	Full-text search of individual functions

Taskbar



CAD Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

1 Workspaces

2 HEROS menu

Further information: "Description of function", Page 663

3 Opened applications, e.g.:

- Control interface
- **CAD Viewer**
- Window of HEROS functions

You can move the opened applications into any other workspaces.

4 Widgets

- Calendar
- Status of the firewall

  Further information: "Firewall", Page 614
- Network status

  Further information: "Ethernet interface", Page 581
- Notifications
- Shut down or restart the operating system

Window Manager

With the Window Manager, you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as **CAD Viewer**.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a taskbar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 82

Notes

- If a window is opened in the third desktop, the control displays an icon in the information bar.

  Further information: "Areas of the control's user interface", Page 82
- The machine manufacturer determines the scope of function and behavior of the window manager.
- The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

26.3 Serial data transfer

Application

The TNC7 automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 626

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: **BAUD_110**, **BAUD_150**, **BAUD_300 BAUD_600**, **BAUD_1200**, **BAUD_2400**, **BAUD_4800**, **BAUD_9600**, **BAUD_19200**, **BAUD_38400**, **BAUD_57600**, **BAUD_115200**
protocol (no. 106702)	Data transfer protocol ■ **STANDARD**: Standard data transfer, line-by-line ■ **BLOCKWISE**: Packet-based data transfer ■ **RAW_DATA**: Transfer without protocol (purely character-by-character) Input: **STANDARD**, **BLOCKWISE**, **RAW_DATA**
dataBits (no. 106703)	Data bits in each transferred character Input: **7 Bit**, **8 Bit**
parity (no. 106704)	Parity bit used to check for transmission errors ■ **NONE**: No parity, no error detection ■ **EVEN**: Even parity, error if the number of bits set is odd ■ **ODD**: Odd parity, error if the number of bits set is even Input: **NONE**, **EVEN**, **ODD**
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize to each transmitted character during serial data transmission. Input: **1 Stop-Bit**, **2 Stop-Bits**
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. ■ **NONE**: No data-flow check ■ **RTS_CTS**: Hardware handshaking, transmission stop is active through RTS ■ **XON_XOFF**: Software handshaking, transmission stop is active through DC3 Input: **NONE**, **RTS_CTS**, **XON_XOFF**
fileSystem (no. 106707)	File system for the serial interface ■ **EXT**: Minimum file system for printers or non-HEIDENHAIN transmission software ■ **FE1**: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: **EXT**, **FE1**

Machine parameters	Setting
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ **TRUE**: The BCC does not correspond to any control character ■ **FALSE**: Function not active Input: **TRUE**, **FALSE**
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state. ■ **TRUE**: Level is **LOW** in idle state ■ **FALSE**: Level is **HIGH** in idle state Input: **TRUE**, **FALSE**
noEotAfterEtx (no. 106710)	This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text). ■ **TRUE**: The EOT character is not sent ■ **FALSE**: The EOT character is sent Input: **TRUE**, **FALSE**

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

Parameter	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transfer protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Type of handshake	RTS_CTS
File system for file operations	FE1

TNCserver is part of the TNCremo software for PCs.

Further information: "PC software for data transfer", Page 671

26.4 PC software for data transfer

Application

HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Related topics

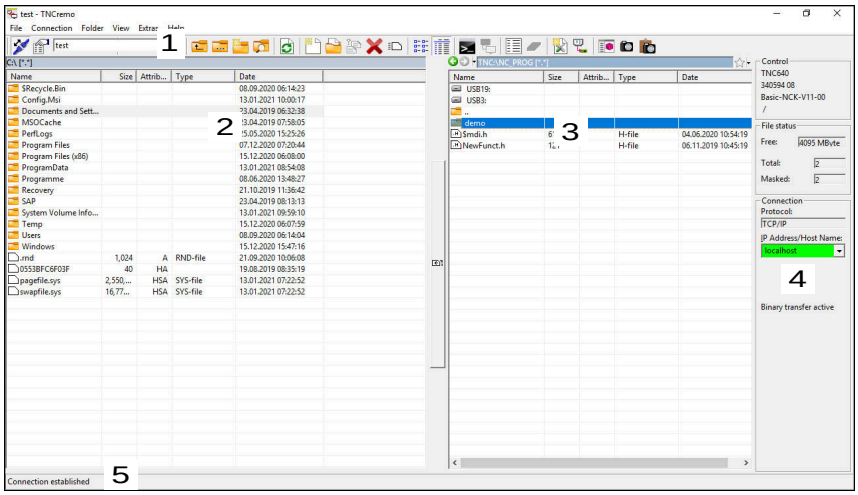
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 678

Requirements

- PC operating system:
  - Windows 8
  - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- A network connection to the control

Description of function

The TNCremo data transfer software provides the following areas:



1 Toolbar

This area provides the most important TNCremo functions.

2 File list of PC

In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).

3 File list of control

In this area, TNCremo displays all of the folders and files of the connected drive of the control.

4 Status display

In the status display, TNCremo shows information about the current connection.

5 Connection status

The connection status indicates whether a connection is currently active.

For more information, refer to the integrated help system of TNCremo.

You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

  If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- You can download the current version of the TNCremo software from the **HEIDENHAIN website** for free.

26.5 File transfer with SFTP (SSH File Transfer Protocol)

Application

SFTP (SSH File Transfer Protocol) provides a secure way to connect client applications to the control and to transfer files at high speed from a PC to the control. The connection is routed via an SSH tunnel.

Related topics

- User administration
  Further information: "User administration", Page 635
- Principle of the SSH connection
  Further information: "Concept of transmission through an SSH tunnel", Page 659
- Firewall settings
  Further information: "Firewall", Page 614
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 678

Requirements

- PC software TNCremo with version 3.3 or higher is installed
  Further information: "PC software for data transfer", Page 671
- **SSH** service is permitted in the firewall of the control
  Further information: "Firewall", Page 614

Description of function

SFTP is a secure transmission protocol supported by various operating systems for client applications.

To set up the connection, you need a key pair consisting of a public and a private key. You transfer the public key to the control and assign it to a user through the user administration. The private key is required by the client application to set up a connection to the control.

HEIDENHAIN recommends using the CreateConnections application to generate the key pair. CreateConnections is installed together with the PC software TNCremo with version 3.3 and higher. CreateConnections lets you transfer the public key directly to the control and assign it to a user.

You can also use other software to generate the key pair.

26.5.1 Setting up an SFTP connection with CreateConnections

For an SFTP connection using CreateConnections, the following are required:

- Connection with secure protocol, such as **TCP/IP Secure**
- User name and password of the desired user are known

> When you transfer the public key to the control, you must enter the user's password twice.
>
> If user administration is inactive, the user **user** is logged in. The password for the user **user** is **user**.

To set up an SFTP connection:

- Select the **Settings** application
- Select **Network/Remote Access**
- Select **DNC**
- Activate the **Setup permitted** toggle switch
- Create a key pair with CreateConnections and transfer it to the control

> For more information, refer to the integrated help system of TNCremo.
>
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

- Deactivate the **Setup permitted** toggle switch
- Transfer the private key to the client application
- Connect the client application to the control

> Please refer to the manual of the client application.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- During the connection, the rights of the user to whom the used key is assigned are active. The directories and files displayed, as well as the access options, vary depending on the permissions.
- You can also transfer a public key to the control by using a USB device or network drive. In this case, you do not need to activate the **Allow password authentication** check box.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

26.6 Secure Remote Access

Application

Secure Remote Access (SRA) allows you to set up an encrypted connection between a PC and your control via the Internet. SRA allows the control to be displayed and operated on a PC, such as for service trainings or remote maintenance.

Related topics

- VNC settings
  Further information: "The VNC menu item", Page 605

Requirements

- Existing Internet connection
  Further information: "Network configuration with Advanced Network Configuration", Page 681
- The following settings in the **VNC settings** window:
  - **Enable RemoteAccess and IPC** check box is active
  - In the **Enabling other VNC** area, the **Inquire** or **Permitted** check box is active

  Further information: "The VNC menu item", Page 605
- PC with paid RemoteAccess software including the extension **Secure Remote Access**

  HEIDENHAIN website

> For more information, refer to the integrated help system of RemoteAccess.
>
> You can open the context-sensitive help function of the RemoteAccess software by pressing the **F1** key.

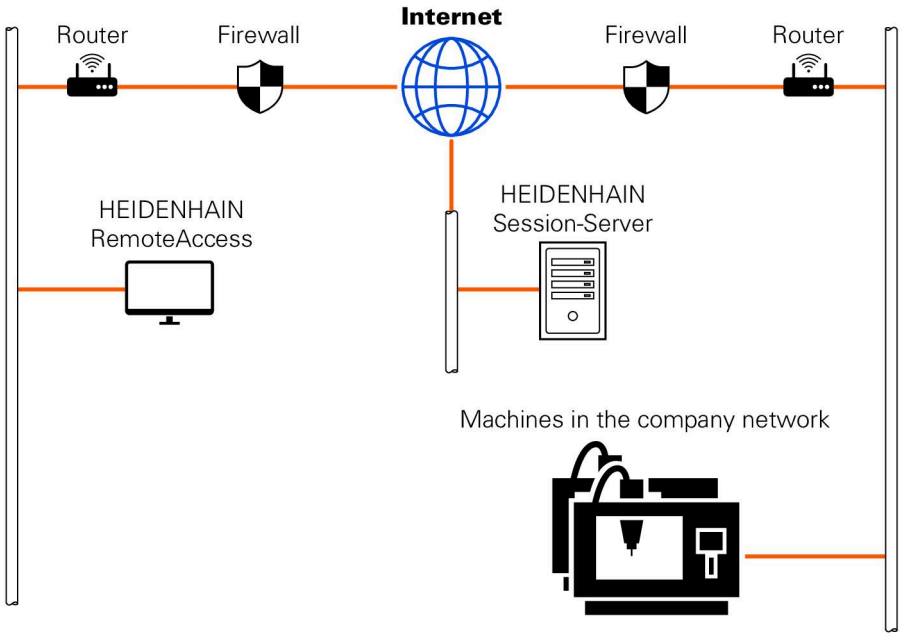
Description of function

To navigate to this function:

Tools ▶ **Secure Remote Access**

The PC provides a ten-digit session ID for you to enter in the **HEIDENHAIN Secure Remote Access** window.

SRA enables connection via an VPN server.



In the **Extended** area, the control shows the progress of the connection setup.

The **HEIDENHAIN Secure Remote Access** window provides the following buttons:

Button	Function
Connect	The control starts the connection with the entered session ID.
Update	The control manually searches for updates for SRA. The control automatically searches for available updates when you open the **HEIDENHAIN Secure Remote Access** window. If an update is available, you can install it. The control restarts during the update.
Config.	The control opens the **Network settings** window. Only for network specialists
Show log	The control opens the log files of the SRA.

Notes

If, in the **VNC settings** window, you set the **Enabling other VNC** setting to **Inquire**, you can permit or deny any connection.

26.7 Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

- File management
  Further information: Programming and Testing User's Manual

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 620

You have the following options for transferring files from the control:

- TNCremo
  With TNCremo you can transfer files from the control to a computer.
  Further information: "PC software for data transfer", Page 671
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 577
- External data carriers
  You can back-up files to external data carriers or use external data carriers to transfer the files.
  Further information: Programming and Testing User's Manual

Notes

- You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.
- You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.
- Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.
- Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.
- HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

26.7.1 Example: Transmission duration of different transmission types

The following table shows the measurement results regarding how long the different services take to transmit an NC program with a size of 1 GB to and from the control.

A CAD laptop and a TNC7 with an MC 356 main computer were used for the test.

Service	Duration of transmission to the control	Duration of transmission from the control	Further information
TNCremo v3	Approx. 3 min 30 s	Approx. 14 min 30 s	Page 671
HEIDENHAIN DNC	Approx. 0 min 12 s	Approx. 0 min 12 s	Page 598
OPC UA NC Server	Approx. 0 min 27 s	Approx. 0 min 50 s	Page 590
SFTP	Approx. 0 min 09 s	Approx. 0 min 10 s	Page 673
SMB	Approx. 0 min 12 s	Approx. 0 min 09 s	Page 581

The duration of transmission can increase due to factors, such as:

- Number of simultaneously transmitted files
- Network utilization
- Age of the hardware being used

26.8 Opening files with additional software

Application

The control provides various tools for opening and editing standard file types.

Related topics

- File types
  Further information: Programming and Testing User's Manual
- Opening image, PDF, and HTML files in the **Document** workspace
  Further information: Programming and Testing User's Manual
- Opening text files in the **Text editor** workspace
  Further information: Programming and Testing User's Manual
- Opening 3D models in **CAD Viewer**
  Further information: "Opening CAD files with CAD Viewer", Page 337

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS) CSV	Gnumeric
INI A TXT CFG	Leafpad
CFG	Combine fixtures or KinematicsDesign **Further information:** "Combining fixtures in the New Fixture window", Page 271
CFT	ToolHolderWizard **Further information:** "Customizing tool carrier templates with ToolHolderWizard", Page 203
HTM/HTML	Web Browser or Mozilla Firefox For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto Image Viewer or Geeqie Ristretto can only open graphics files. Geeqie can also edit and print graphics.

File type	Tool
OGG	Parole

> With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file in the correct workspace or tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

26.8.1 Opening tools

To open a tool:

- ▶ Select the HEIDENHAIN icon in the taskbar
- > The control opens the HEROS menu.
- ▶ Select **Tools**
- ▶ Select the tool (e.g. **Leafpad**)
- > The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.

  If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

  Further information: "SELinux security software", Page 576

26.9 Network configuration with Advanced Network Configuration

Application

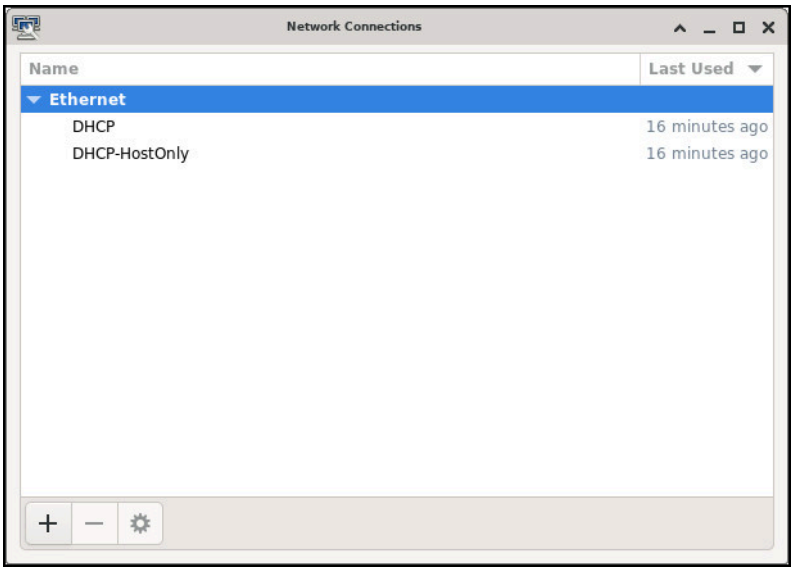
Use **Advanced Network Configuration** to edit or remove profiles for the network connection.

Related topics

- Network settings
  Further information: "The Editing network connection window", Page 682

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.



The **Network Connections** window

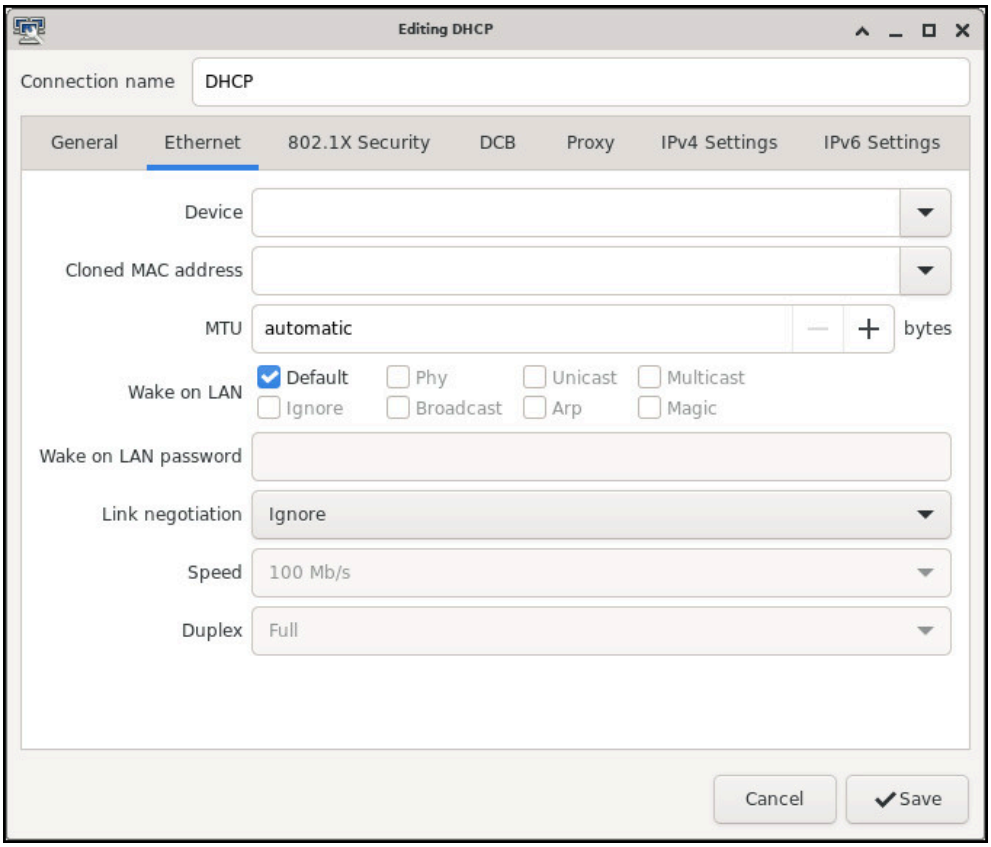
Icons in the Network Connections window

The following icons are shown in the **Network Connections** window:

Icon	Function
+	Add network connection
−	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "The Editing network connection window", Page 682

26.9.1 The Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.



The **Editing network connection** window

The General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically	If you are using several profiles, you can define an order of priority for the connection here. The control connects the network with the highest priority first. Input: **-999...999**
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN when using this connection	Currently no function
Bonded connections:	Currently no function

The Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Service:	Here you can select the Ethernet interface. If you do not select an Ethernet interface, this profile can be used for any Ethernet interface. Selection by means of a selection window
Cloned MAC address:	Currently no function
MTU:	Here you can define the maximum package size in bytes. Input: **Automatic**, **1...10000**
Private key password:	Currently no function
Wake-on-LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection: ■ **Ignore** Retain the configurations already existing on the device. ■ **Automatic** The speed and duplex settings are configured automatically for the connection. ■ **Manual** Configure the speed and duplex settings for the connection manually. Selection by means of a selection window
Speed	Here you have to select the speed settings: ■ **10 Mb/s** ■ **100 Mb/s** ■ **1 Gb/s** ■ **10 Gb/s** Only if **Link negotiation** has been selected **Manual** Selection by means of a selection window
Full duplex	Here you have to select the duplex setting: ■ **Half** ■ **Full** Only if **Link negotiation** has been selected **Manual** Selection by means of a selection window

The 802.1X Security tab

Currently no function

The DCB tab

Currently no function

The Proxy tab

Currently no function

The IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
Method:	Here you have to select a network connection method: ■ **Automatic (DHCP)** If the network uses a DHCP server for IP address assignment ■ **Automatic (DHCP) addresses only** If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually ■ **Manual** Assign the IP address manually ■ **Link-Local Only** Currently no function ■ **Shared to other computers** Currently no function ■ **Disabled** Deactivate IPv4 for this connection
Automatic, addresses only	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with **Method: Manual**
Additional DNS servers:	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with **Method: Manual** and **Automatic (DHCP) addresses only**
Additional search domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **Method: Manual**
DHCP client ID:	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

The IPv6 Settings tab

Currently no function

27

Overviews

27.1 Pin layout and cables for data interfaces

27.1.1 V.24/RS-232-C interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50178 for Secure separation from the power grid.

Control		25-pin: VB 274545-xx			9-pin: VB 366964-xx		
Male	**Assignment**	**Male**	**Color**	**Female**	**Female**	**Color**	**Female**
1	Do not assign	1	White/Brown	1	1	Red	1
2	RXD	3	Yellow	2	2	Yellow	3
3	TXD	2	Green	3	3	White	2
4	DTR	20	Brown	8	4	Brown	6
5	Signal GND	7	Red	7	5	Black	5
6	DSR	6		6	6	Violet	4
7	RTS	4	Gray	5	7	Gray	8
8	CTR	5	Pink	4	8	White/Green	7
9	Do not assign	8	Violet	20	9	Green	9
Housing	External shield	Housing	External shield	Housing	Housing	External shield	Housing

27.1.2 Ethernet interface RJ45 socket

Maximum cable length:

- 100 m unshielded
- 400 m shielded

Pin	Signal
1	TX+
2	TX–
3	RX+
4	Vacant
5	Vacant
6	RX–
7	Vacant
8	Vacant

27.2 User administration roles and rights

27.2.1 List of roles

The following contents can change in the following software versions of the control:

- HEROS role names
- Unix groups
- Basic ID number

Further information: "Roles", Page 638

Operating system roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 335
	■ HEROS.Printer	■ lp	■ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	■ HEROS.SetShares	■ mntcfg	■ 334
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
	■ HEROS.MountUSBDevices	■ mntusb	■ 345
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 337
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 341
	■ HEROS.SetNetwork	■ netadmin	■ 336
	■ HEROS.SetTimezone	■ tz	■ 333
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.LegacyUserNoCtrlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	■ HEROS.BackupMachine ■ HEROS.UserAdmin	■ backup ■ useradmin	■ 338 ■ 339

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	■ NC.OPModeProgramRun ■ NC.OpModeSingleStep	■ NCOpPgmRun ■ NCOpSinglesStep	■ 302 ■ 303
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	■ NC.EditNCProgram ■ NC.EditPalletTable ■ NC.EditPresetTable ■ NC.EditToolTable ■ NC.OPModeMDi ■ NC.OPModeManual	■ NCEdNCProg ■ NCEdPal ■ NCEdPreset ■ NCEdTool ■ NCOpMDI ■ NCOpManual	■ 305 ■ 309 ■ 308 ■ 306 ■ 301 ■ 300
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the Programmer role, as well as the following rights:		
	■ NC.ApproveFsAxis ■ NC.EditPocketTable ■ NC.SetupDrive ■ NC.SetupProgramRun	■ NCApproveFsAxis ■ NCEdPocket ■ NCSetupDrv ■ NCSetupPgRun	■ 319 ■ 307 ■ 315 ■ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	■ NC.ScheduleProgramRun	■ NCScheduleProgRun	■ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter. Exception: The **Legacy-User** does not have the NC.ApproveFsAxis right.		

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors. ■ Special functions for the programming of variables and for editing the table header Replacement for code number **555343**		
	■ NC.EditNCProgramAdv ■ NC.EditTableAdv	■ NCEdit-NCPgmAdv ■ NCEdit-TableAdv	■ 327 ■ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	■ NC.RemoteProgramRun	■ NCRe-motePgmRun	■ 329

Machine manufacturer (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	■ NC.ConfigUserAdv ■ NC.SetupDrive	■ NCConfi-gUserAdv ■ NCSetupDrv	■ 316 ■ 315
PLC.ServiceRead	This role allows read-only access during servicing. This role can be used to display various types of diagnostic information		
	■ NC.Data.AccessServiceRead	■ NCDASer-viceRead	■ 324

Refer to your machine manual.

The machine manufacturer can adapt the PLC roles.

When the **Machine manufacturer (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:

- The names of the roles
- The number of roles
- The functionality of the roles

27.2.2 List of rights

The table below lists all of the individual rights.

Further information: "Rights", Page 638

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operation of the machine in the **Manual** operating mode
NC.OPModeMDi	Working in the **MDI** application
NC.OpModeProgramRun	Running NC programs in Full Sequence mode of the **Program Run** operating mode
NC.OpModeSingleStep	Running NC programs in **Single Block** mode of the **Program Run** operating mode
NC.SetupProgramRun	Probing in the **Setup** application Using the **AFC** and **ACC** functions
NC.ScheduleProgramRun	Programming a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes Resetting the test position of the axes
NC.EditNCProgramAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Setting the date and time, time zone and time synchronization via NTP and the **HEROS menu**
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.MountUSBDevices	Access to USB devices (e.g., USB flash drive, external hard disk) via the file manager
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration

HEROS role name	Description
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	Defining the NC program start and override values from an external application (e.g., via the DNC interface) Read-access and write-access to the counter by means of **FUNCTION COUNT** via **OPC UA NC Server** (#56-61 / #3-02-1*)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.DataAccessServiceRead	Read-only access to the **PLC:** drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through **OPC UA NC Server** (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaOEMConfiguredData	Read-access and write-access through **OPC UA NC Server** (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaPwAuth	Logon to **OPC UA NC Server** (#56-61 / #3-02-1*) with your user name and password
NC.OpcUaPwAuthOnlyMachineNet	For connection via the **eth1** network interface: Logon to **OPC UA NC Server** (#56-61 / #3-02-1*) with your user name and password

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
Klartext on your mobile device

Google Play Store

Apple App Store

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:
www.heidenhain.com/products/touch-probes-and-vision-systems

Original document
1358774-23 · Ver03 · 09/2024 · H · Printed in Germany