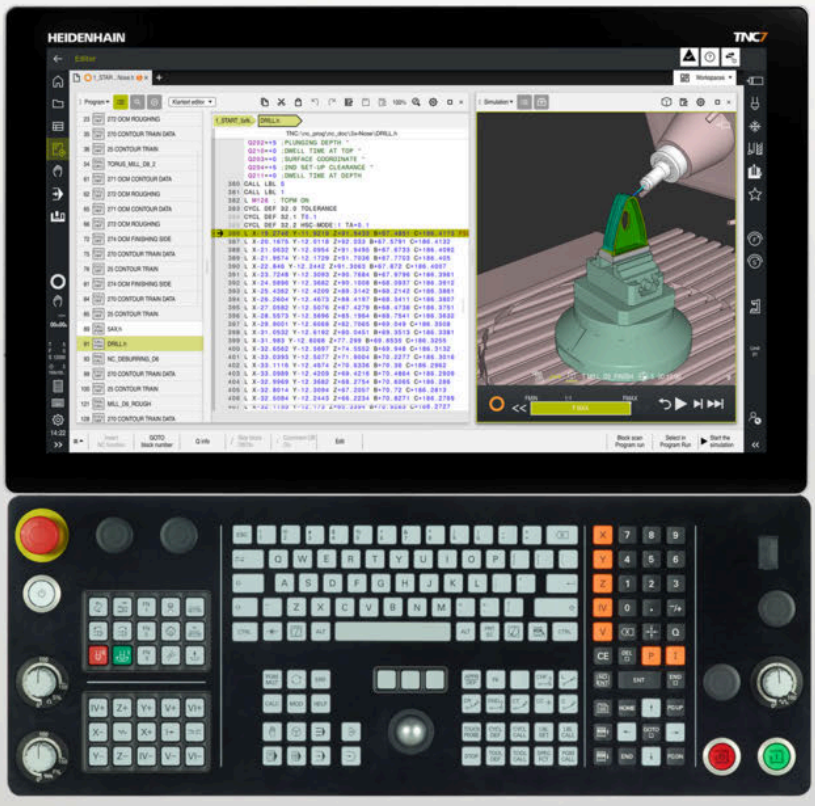
HEIDENHAIN

TNC7

User's Manual
Setup and Program Run

NC Software
81762x-20



English (en)
10/2025

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.

https://content.heidenhain.de/doku/tnc_guide/html/en/

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 Process Tracking Interface (#3-04-1*)

Topic	Description
Software option PTI 8 channels (#3-04-1*)	This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with **SIK2**. You can order this software option up to two times, each time selecting up to eight channels for recording. **Further information:** "Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)", Page 714

1.1.2 Dynamic Collision Monitoring DCM (#140 / #5-03-2)

Topic	Description
The Collision Monitoring software option	The software option Collision Monitoring (#40 / #5-03-1) is no longer available. To be able to use the Dynamic Collision Monitoring (DCM) functions in the future, you need the software option Collision Monitoring v2 (#140 / #5-03-2) If the software option Collision Monitoring v2 (#140 / #5-03-2) is not available after a software update, the control will display a warning.

1.1.3 About the product

Topic	Description
Workspaces	When you select the title of a workspace, the control will open a selection menu. This selection menu allows you to switch workspaces without changing their size or position. You can only switch between workspaces that can be closed. **Further information:** "Operating elements within the workspaces", Page 92
Customizing the user interface	The machine manufacturer can use the machine parameter **state** (no. 143601) to dim or hide the following user interface elements, for example: ■ Operating modes ■ Applications ■ Workspaces ■ Buttons Thus, it is possible to remove certain areas from the control's user interface for training purposes. Refer to your machine manual.
	If the machine is equipped with the required sensor technology, the machine manufacturer can set up a Resource Usage Monitor (RUM). In RUM, usage of various supplies can be displayed, such as electricity or compressed air. Refer to your machine manual.

1.1.4 Status displays

Topic	Description
Position display mode	The **Actual distance to go (ACTDST)** and **Nominal distance to go (REFDST)** modes have been added to the position display: ■ **Actual distance to go (ACTDST)** displays the distance-to-go of the programmed axes in the input coordinate system **I-CS**. ■ **Nominal distance to go (REFDST)** displays the distance-to-go of the physical axes in the machine coordinate system **M-CS** that are required for the programmed traverse. **Further information:** "Position displays", Page 158
The **Status** workspace	On the **PROCMON** tab of the **Status** workspace, the **Process monitoring enforced reactions** (#168 / #5-01-1) area was added. The area contains a table with all reactions that have been triggered since the control was switched on. You can filter the contents of this table. **Further information:** "The PROCMON (#168 / #5-01-1) tab", Page 148
	On the **COMPMON** tab of the **Status** workspace, the **Enforced reactions of component monitoring** (#155 / #5-02-1) area was added. The area contains a table with all reactions that have been triggered since the control was switched on. You can filter the contents of this table. **Further information:** "The COMPMON (#155 / #5-02-1) tab", Page 139

1.1.5 Tools

Topic	Description
Tool type	The **Calibration mandrel** (**CAL_PIN**) tool type was added. You can use the **Calibration mandrel** to calibrate the tool touch probe with Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**). **Further information:** "Tool types", Page 200

1.1.6 Process monitoring (#168 / #5-01-1)

Topic	Description
Start page	If process monitoring has not been activated or was reset for an NC program, the control will display a start page in the **Process Monitoring** workspace. The control displays a table with all available monitoring tasks in rows and possible process-monitoring use cases in columns. The stars in the cells indicate how well suited the monitoring task is for the corresponding use cases. The same table will be displayed each time you change the monitoring task in process monitoring. **Further information:** "Process monitoring start page", Page 317
Signals	With the **Unexpected servo lag** signal, the control can monitor machining for tool or cutting edge breakage. For this signal, the control offers the following monitoring tasks: ■ **Unexpected servo lag — Constant** (active by default) ■ **Unexpected servo lag — Tunnel** **Further information:** "Signals", Page 310
	For machines with GEN3 drives, the **Torque** signal was added. Process monitoring calculates the spindle torque, which possibly allows more precise analyses than the spindle current. For this signal, the control offers the following monitoring tasks: ■ **Torque — Waveform Comparison** (active by default) ■ **Torque — Tunnel** ■ **Torque — Constant** ■ **Torque — Display**
Spindle current — Constant monitoring task	The **Spindle current — Constant** monitoring task allows you to collect recordings of the reference signal, while monitoring it right from the first machining operation.
Form in the **Setup** mode	In the **Parameter settings** area, the control will display all monitoring tasks that have been marked in the **Setup** area as tabs. Each tab displays the parameters of the corresponding monitoring task. You can change the parameterization using sliders. **Further information:** "The Form column in Setup mode", Page 325
Form in the **Runtime** mode	In the **Program-run notifications** area, you can use toggle switches to determine whether the control will display or suppress certain warning messages for this NC program.
	In the **Triggered reactions** area, the control will display all triggered reactions for the recordings currently marked in the **Recordings - Table** area. **Further information:** "The Form column in Runtime mode", Page 329
Graphs for monitoring tasks	The **Intersecting lines** icon was added to the graphical representation of the monitoring task. It can be used to display vertical lines. The control will mark the intersections of the graphs with the lines. Next to the intersections, the control displays the resulting value color or the numerical values of the signal, depending on the active **Signal display**. **Further information:** "Graphic display of the monitoring tasks", Page 332

Topic	Description
The **Settings** window	In the **Recordings - Table** area, you can define the number of decimal places **0...8** for the display of floating-point numbers in the table. **Further information:** "The Settings window", Page 336
The **Recordings - Graph** area	In the **Recordings - Graph** area, the control by default shows the history of the resulting value across the existing recordings. You can change the displayed values (for example, in order to detect correlations or trends). **Further information:** "The Recordings - Graph area", Page 337
Parameter learning	The parameter learning function allows you to set the Process Monitoring parameters without knowing the required values. You can define the maximum deviation of the machining operation from the error limit. The control will use this information to calculate the parameter values. This function can also be used during machining. If you use a monitoring task with the **Constant** method, you can monitor even the first machining operation based on a parameterization matching the workpiece. **Further information:** "Parameter learning", Page 344

1.1.7 CAD Viewer

Topic	Description
The **Hover** mode	The **Hover** icon was added to replace the **Rotate** mode. If **Hover** is active and you swipe across the CAD model during touch operation, the control will display selectable options in orange. The function is comparable to hovering with a mouse. As soon as you stop the movement, the control will mark the current selection option. If **Hover** is active, you can no longer rotate the CAD model. **Further information:** "Screen layout", Page 351
Loading machining positions (#42 / #1-03-1)	Using the **H file with FMAX M99 at each position (for older NC controls)** icon, you can determine whether the control adds a cycle call for each position in the NC program or not. **Further information:** "Applying positions", Page 364

1.1.8 Program Run

Topic	Description
The **Manual traverse** function	Using the optional machine parameter **userControl** (no. 144101), the machine manufacturer defines if the control will display the **Handwheel** toggle switch or not. **Further information:** "Manual traverse during an interruption", Page 437

1.1.9 Tables

Topic	Description
Table filters	The control provides the **Calibration tools** tool filter.
Machine manufacturer settings	Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus. Refer to your machine manual.

1.1.10 The Settings application

Topic	Description
Code numbers	The **Home** operating mode icon shown in the TNC bar changes if the code number 123 is enabled. The icon additionally shows an unlocked padlock to illustrate that machine parameters can be accessed. **Further information:** "Code numbers", Page 580
The **Machine Settings** menu item	If the machine manufacturer has enabled the function, you can define the operating station active by default for movements during a program-run interruption. **Further information:** "The Machine and Simulation areas", Page 581
The **Expanded logging** menu item	The **Expanded logging** menu item has been added to replace the optional machine parameter **extendedDiagnosis** (no. 124204). If this function is active, the control will save graphics journal data. These data may be needed by the HEIDENHAIN Customer Service for diagnostic purposes. If you activate this function, you need to restart the control. Deactivate this function after the fault has been fixed to reduce the amount of memory used.

1.1.11 Machine parameters

Topic	Description
Open context menu	For all objects in the configuration editor, the control provides an icon that opens the context menu. Objects are shown by a folder icon.
The **Highlight change** button	The **Highlight change** button has been added to the function bar of the configuration editor. The control marks the selected machine parameter as changed, even though the value remains the same. The machine manufacturer can use this function for testing, for example. **Further information:** "Icons, buttons and shortcuts", Page 648
Read-only NC programs	Using the optional machine parameter **editWhileProtected** (no. 105416), you can define whether it is possible to edit a read-only file. By default, you can edit such a file, but only save the changes in a new file with **Save as**. If you set this machine parameter to **FALSE**, the control will dim any editing functions.
Tool management	With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters. **Further information:** "Tool management ", Page 203
Expanded logging	The optional machine parameter **extendedDiagnosis** (no. 134204) has been removed. Instead, you can activate the saving of graphics journal data in the **Expanded logging** menu item of the **Settings** application. **Further information:** "Overview", Page 576

1.1.12 HEROS operating system

Topic	Description
HEROS menu	The HEROS tool **Virtual keyboard** was added. The virtual keyboard is also available if only the operating system is active and the NC software is deactivated.

1.2 Modified or extended functions

1.2.1 About the product

Topic	Description
Workspaces	As an alternative to using the **Move** icon in the title bar, you can move workspaces by dragging the title area.
Scaling windows	If a window is displayed in the right half of the screen, the control displays the scaling icon in the left corner of the window instead of the right corner. **Further information:** "Icons on the control's user interface", Page 105
The **Start** workspace	The **Start/Login** workspace was renamed **Start**.
	Once startup has been completed, the control will close the **Start** workspace. **Further information:** "Powering on and off", Page 161

1.2.2 Status displays

Topic	Description
The **Positions** workspace	If dressing mode **FUNCTION DRESS** is active, the control shows a corresponding symbol in the **Positions** (#156 / #4-04-1) workspace. **Further information:** "Active functions", Page 130
The **Status** workspace	On the **Tool** tab, the control shows the tool life in minutes instead of in hours and minutes. This format is also used in tool management. **Further information:** "The Tool tab", Page 152
	On the **Tool** tab, the information about the tool axis and tool type has been integrated in the **Tool information** area. There is no longer a separate **Tool type** area.
	On the **Tool** tab, the control displays specific tool parameters for grinding tools (#156 / #4-04-1) in the **Tool information** and **Tool allowances** areas.
	On the **Tool** tab, the control displays information about grinding and dressing tools (#156 / #4-04-1), depending on the tool orientation **TO** and the active grinding wheel edge. The control indicates the active grinding wheel edge by a red dot in the tool type graphic.
The **Simulation status** workspace	The following contents were removed from the **POS** tab of the **Simulation status** workspace: ■ Active **Feed** in mm/min ■ Active **Feed-rate override** in % ■ Active **Spindle override** in % **Further information:** "The POS tab", Page 145

1.2.3 Powering on and off

Topic	Description
Configuration files	During booting, the control checks whether all the needed configuration files are available in the **TNC:\config** folder and on the **HOME:** drive. If a file is missing, the control recreates the file and resumes booting.

1.2.4 Manual operation (#50 / #4-03-1)

Topic	Description
Measure unbalance	If the Inch unit of measure is active in the control, the radial position will be displayed in inches and the mass in pounds in the **Result diagram** window.

1.2.5 Tools

Topic	Description
Grinding tools (#156 / #4-04-1)	With the **V_MAX** parameter, you can define the maximum allowed cutting speed in addition to the maximum speed indicated with **NMAX**. The control will use the smaller of both values as the limit. **Further information:** "Parameters of the grinding tool table toolgrind.grd", Page 495
Tool management	If a tool has been saved to the pocket table, you cannot reset the corresponding row in tool management. The tool must be removed from the magazine first.

1.2.6 Collision monitoring (#140 / #5-03-2)

Topic	Description
Set up fixtures	For the selection of the 3D model, the **Last files** area has been added. The control displays the most recently selected 3D models of the file types ***.stl** and ***.cfg**. **Further information:** "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258

1.2.7 Process monitoring (#168 / #5-01-1)

Topic	Description
MONITORING SECTION	Within the **MONITORING SECTION** NC function, the **PAUSE** and **RESUME** syntax elements were added. With **MONITORING SECTION PAUSE** and **MONITORING SECTION RESUME**, you can exclude parts of a monitoring section from process monitoring. The control will neither calculate any resulting values nor trigger reactions. This way, you can exclude a part of the monitoring section that cannot be reproduced without having to delete the recordings previously made.
Changes in the NC program	If your changes to a monitored NC program are restricted to variables, you can keep the previous recordings.
Display of Process Monitoring	The **Process Monitoring** workspace was redesigned. The contents have been grouped into areas that can be collapsed and expanded individually.
General area	You can activate or deactivate process monitoring by means of the toggle switch in the general area of the workspace while program run is interrupted or stopped. **Further information:** "General area", Page 320
	If you select the **Reset** icon, the control will open a window with a confirmation prompt. There, you can specify which of the following settings are to be reset to their defaults: ■ Selected monitoring tasks ■ Columns in the **Recordings - Table** area ■ Recording strategy
The **Filters** column in **Setup** mode	The **Section search** area was added to the **Filters** column in **Setup** mode. The control displays the monitoring sections whose names contain the search term. You can use regular expressions for this purpose. **Further information:** "The Filters column in Setup mode", Page 324
The **Recordings - Table** area in **Runtime** mode	Both the left and the right part of the table can accommodate up to six columns. **Further information:** "The Recordings – Table area", Page 326
	Next to the area title, the control displays in parentheses how many rows the table contains. If a filter is active, the control shows the number of the currently displayed recordings in the **(x/x)** format.
	If program run was aborted, the control will display the resulting value in gray.
The **Form** column in **Runtime** mode	The order of the areas in the **Form** column was changed. The **Recording strategy** and **Evaluate component** areas were swapped. **Further information:** "The Form column in Runtime mode", Page 329
	Strategies for overwriting files were added to the **Recording strategy** area.

Topic	Description
	Once the maximum disk space is used up or the limit you have set is reached, the control will overwrite recordings using one of the following strategies: ■ **Newest recording** ■ **Oldest not-evaluated recording** ■ **Never** The first recording will never be overwritten.
Parameterization of monitoring tasks	During the parameterization of monitoring tasks, the control displays the number of available **Parameter settings** and **Reactions** within parentheses.
	Within the settings for monitoring task parameterization, the left area is collapsed by default. **Further information:** "Settings for parameterizing of monitoring tasks", Page 335
The **Configure table** window	Instead of tapping or clicking, long press or right-click to open the **Configure table** window. **Further information:** "The Configure table window", Page 328
	The **Configure table** window shows the name of the monitoring section from within which the window was opened.
	In the **Configure table** window you can select the contents for all the columns of a table, and not only for one single column. The window shows the table columns as tabs aligned next to each other. You can drag the tabs representing the table columns to rearrange the order, or you can delete them.
	The **Configure table** window shows icons for each selection option so that you can mark your selection as a favorite, add it to the selected column, or add it as a new column.
	The following selection options have been added: ■ **Recording**: recording number ■ **Possible teach-in cuts for reference**: Number of possible reference machining operations for this monitoring section ■ **Fingerprint of path calculation**: If the control displays a different number, the path calculation settings have changed. This machining operation is possibly no longer comparable with the previous machining operations. ■ Numerical **Statistics** for this signal: ■ **Minimum** ■ **Maximum** ■ **Average** ■ **Peak-to-peak** ■ **Standard deviation** **Further information:** "Overview of the selection options in a table or graph", Page 339
Evaluate component	During program run, you can evaluate the workpiece machined in an existing recording as **Part is OK** or **Part not OK**. This recording will then be used after saving as reference machining operation, starting with the next monitoring section.

1.2.8 User aids

Topic	Description
Context menu	The context menu of the **Document** workspace was removed.
Cutting data calculator	The representation of the cutting data calculator was improved. The cutting data calculator now includes only those input fields that are relevant to the currently active NC function.
Message menu	The message menu was revised. Using the hand pointer, you can enlarge or reduce the size of both areas. **Further information:** "Message menu on the information bar", Page 379
FUNCTION REPORT	The **FUNCTION REPORT** NC function now includes the **ARGS** syntax element for texts in PO files. You can enter additional text that will be displayed in the message details.
The **GOTO** function in the **Program** workspace	In the **GOTO jump instruction** window, the **GOTO 0** and **Select last** buttons were added. With **GOTO 0**, the control jumps to the beginning of the NC program and then closes the window. If the user clicks the **Select last** button in the **Program Run** operating mode, the control will copy the number of the last executed NC block to the **Block number** row.

1.2.9 Touch probe functions in the Manual operating mode

Topic	Description
The **Change the preset** window	The control displays a symbol in front of the row of the current preset, like in the preset table.
	If you apply the changes made in the **Change the preset** window while the active preset is locked, the control will open a window with a confirmation prompt. **Further information:** "The Change the preset window", Page 396
Applying the result	If you use the **Compensate the active preset** or **Correct the pallet reference point** function to write to a locked table row, the control will open a window with a confirmation prompt.
Set up the workpiece (#159 / #1-07-1)	For the selection of the 3D model, the **Last files** area has been added. The control displays the most recently selected 3D models of the file types ***.stl** and ***.cfg**. **Further information:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 411
Log file	The maximum file size of the automatic log file **TCHPRMAN.html** was increased to 512 kB. **Further information:** "Log file of touch-probe functions", Page 397

1.2.10 Program run

Topic	Description
Buttons	The buttons in the function bar of the **Program Run** operating mode will always be displayed, even when no NC program is selected.
The **Program** workspace	If the control takes long for saving in the **Editor** operating mode, it will display a progress bar for this NC program even in the **Program Run** operating mode. Machining cannot be started before the NC program has been saved.

1.2.11 Tables

Topic	Description
The **Table** workspace	In the dialog bar of the **Table** workspace, the control displays the unit of the selected table cell. **Further information:** "The Table workspace", Page 462
	If you insert the contents of the clipboard while editing is inactive, the control displays the **Editing disabled. Enable?** prompt. If you select **Yes**, the control inserts the copied contents.
	The sequence of columns defined in the **Tables** window applies to all filters for the table. If user administration is active, the control stores the column sequence for each user. After updating the software to version 20, you must redefine the column sequence if you changed it.
The **Form** workspace	You can scale the content of the **Form** workspace to make it look larger or smaller. **Further information:** "Icons", Page 471
	In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas. **Further information:** "The Form workspace for tables", Page 470
	For the following tool types, the **CUT** parameter was added for the number of teeth in the **Additional geometry data** area: ■ **Thread mill** (**GF**) ■ **Thread miller with chamfer** (**GSF**) ■ **Thread mill with single thread** (**EP**) ■ **Thread mill w/ indexable insert** (**WSP**) ■ **Circular thread mill** (**ZBGF**)
Tool management	When you open the selection window for the **KINEMATIC** parameter, you can directly enter characters to search for a tool carrier kinematic model.
Grinding tool table (#156 / #4-04-1)	For inch tables, you define the **V** and **V_MAX** parameters in ft/min. **Further information:** "Parameters of the grinding tool table toolgrind.grd", Page 495
Pocket table **tool_p.tch**	If you define a tool in the column **T** of the pocket table, you can enter the tool number directly. The control opens the **Tool selection** window and selects the defined tool. You can load the tool to the column **T**.

Topic	Description
Freely definable tables	When you edit cells of the type **PATHNAME**, the control automatically opens the **Open File** selection window.
Column names	Column names are not case-sensitive. If the only difference in column names is their case, the control will not open the table, but display an error message.
The **Incomplete table layout** window	If you modify a table format in the **Incomplete table layout** window, the control applies the same logic as is used for the **Update TAB / PGM** button.
Exporting table contents	When you export the contents of the **Tooling list** table, the CSV file contains all contents of the tool, and not just the columns of the Tooling list. This allows you to import this CSV file to the tool management. **Further information:** "Importing and exporting of table contents", Page 473
	The pocket table no longer permits the export of table contents.
	The table **T usage order** (#93 / #2-03-1) no longer permits the export of table contents.
Importing table contents	The **Confirm import** window has been revised: ■ The control opens the window for every import. ■ The window shows the entire content to be imported. ■ You can select separately for each row whether the control will append, overwrite, or ignore the content. ■ For example, if you append tool data, you can optionally enter a new tool number. **Further information:** "The Confirm import window", Page 474
Machine manufacturer settings	The optional machine parameter **choice** (no. 105704) for machine manufacturers is now available for additional parameters and tables. The machine manufacturer can thus modify the appearance of toggle switches in the **Form** workspace.

1.2.12 Electronic handwheel

Topic	Description
The **Handwheel** toggle switch	The position of the **Handwheel** toggle switch has been changed in the function bar. The control will display the **Handwheel** toggle switch on the right side of the function bar, next to the **Internal stop** button.

1.2.13 The Settings application

Topic	Description
Enabling software options with **SIK2**	To enable a software option for a control with **SIK2**, you need a key file (from the machine manufacturer, for example). If a key file is present in the **TNC:\SIK2** folder, the control will read that file automatically during the next startup. **Further information:** "Code numbers", Page 580

Topic	Description
The **General Information** menu item	In the **Version Information** area, the control displays the **Operating System** subsection and perhaps also the **OEM cycles** subsection. In the **Machine information** area, the control displays the **Hardware** subsection with the machine's RAM. **Further information:** "The General Information menu item", Page 584
The **Adjustment of analog voltage offset** menu item	The table for analog axes was revised. Instead of the **Offset in mV** column, the table now contains the columns **Current offset**, **New offset**, and **Deviation**. **Further information:** "The Adjustment of analog voltage offset menu item", Page 593
Network drives	You can use the **Single Sign On** setting not only for private networks, but also for all networks. The control uses the respective user's logon data for the public network drive. **Further information:** "Network drives on the control", Page 598
OPC UA connection assistant (#56-61 / #3-02-1*)	The **OPC UA connection assistant** also exports self-generated server certificates. **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 617
The **DNC** menu item	If the **DNC access permitted** toggle switch is inactive, the **TNCopt full access allowed** toggle switch is dimmed instead of hidden. **Further information:** "The DNC menu item", Page 619
Configurations	With the **Enable autosave** toggle switch, you can define whether the control automatically saves changes to the current configuration.
	If you reset or save a configuration, the control displays a confirmation prompt. If you reset to a saved configuration, the control will display the date of saving. **Further information:** "Configuring the control's user interface", Page 653

1.2.14 Machine parameters

Topic	Description
Configuration editor	The search function takes all machine parameters into account, independent of the current cursor position. **Further information:** "Machine parameters", Page 646
	In the **Configuration** window, the **Do not show empty folders** toggle switch was added. This toggle switch can be used to hide empty attributes or folders.

1.2.15 User administration

Topic	Description
The **Active user** window	The control displays the assigned roles in the **Active user** window. **Further information:** "The Active user window", Page 665

1.2.16 HEROS operating system

Topic	Description
Screensaver	The **GLSlideshow** was added. You can select a folder that contains JPG or PNG graphics to be displayed by the control.
NC/PLC Backup and **NC/PLC Restore**	You can create and restore backups of more than 4 GB in file size.

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., comprehending 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 PDF file containing a **complete edition** of the User's Manual with all contents

  ID: 1369999-xx

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- 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 executing 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 machining cycles functions.

    ID: 1358775-xx
  - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions related to the touch-probe cycles.

    ID: 1358777-xx

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 66

Complete edition and TNCguide for the TNC7

Binding of the printed version of the User’s Manual is only possible if a certain number of pages is not exceeded.

Die **complete edition** and **TNCguide** may include additional sample programs for NC functions that are not contained in the printed version.

You can download this documentation free of charge from the HEIDENHAIN website.

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.

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- **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

  https://content.heidenhain.de/doku/tnc_guide/html/en/
- 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

Caution indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, such as 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 53

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 66

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

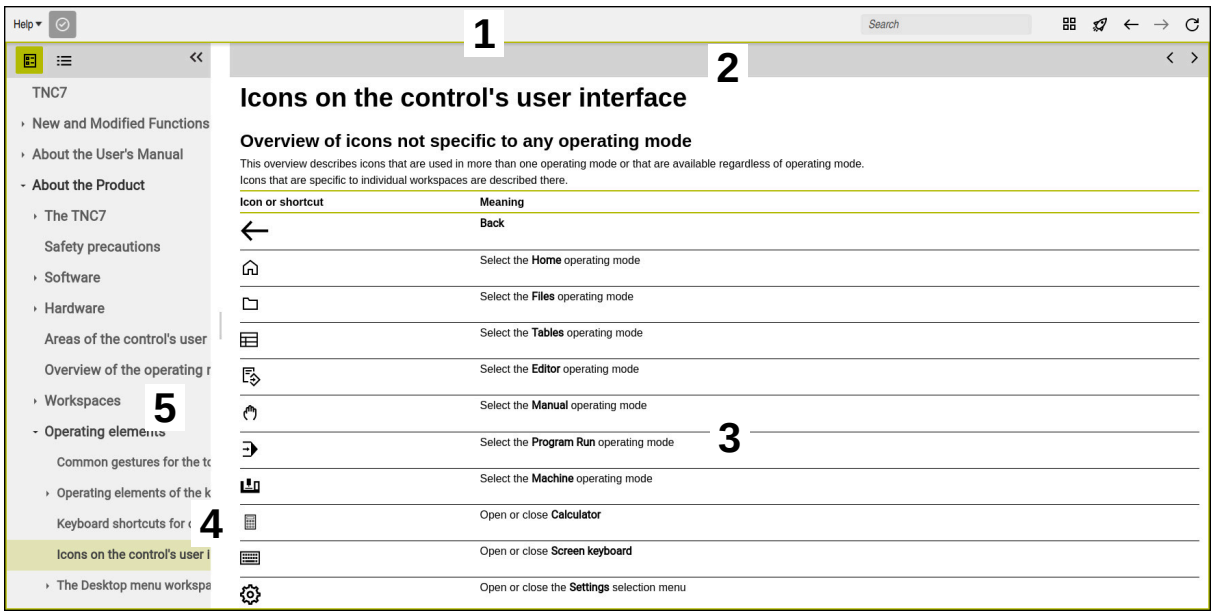
Further information: "The Help application", Page 59

Further information: Programming and Testing User's Manual

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

Further information: "Icons", Page 60

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 60

2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 60

3 Content column of the **TNCguide**

4 Separator between the columns of the **TNCguide**
Adjust the column width by means of the separator.

5 Navigation column of the **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 61
	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 62

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 59

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 57

- 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

Would you like any changes, or have you found any errors?

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.com

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 86

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 customer feedback into consideration. This results, for example, in function adaptations of the controls and safety precautions in the information products.

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

Further information: "Contacting the editorial staff", Page 63

3.1.2 Data Architecture and Access Options

Directories

While in use, the control will store various types of data on its internal data carrier.

Partition or directory	Data
root	Operating system
/mnt/tnc/	Process, application, and user data Examples: NC programs, tables or 3D models
/mnt/plc/	Machine manufacturer data Examples: PLC program, machine configuration or additional software
/mnt/sys/	NC software

The type, format, and amount of data stored depend on various factors. For example, the software options that are available and actually used as well as the functions used, including the current settings, have a great influence here.

Data acquisition

While in use, the control will generate different pieces of data. Depending on the functions in use, for example, data acquisition is performed continuously and in real time.

Data acquisition examples:

- Backup

  You can manually create a backup copy of the TNC partition in order to avoid potential loss of data.
- Service file

  In the event of a fault, you can manually create a service file that supports troubleshooting by a service technician.
- Process monitoring (#168 / #5-01-1)

  The control compares, for example, recorded and current process data in order to trigger defined reactions.
- User administration

  If user administration is active, the control assigns defined roles and permissions to the logged-on users. When a user logs on, the control will acquire personal or anonymous data in connection with its use.
- Tool-usage file

  Using tool-usage files, the control can, for example, check whether the tool's remaining life is sufficient for the current NC program.
- Log functions or log files (e.g., from touch-probe cycles)

  Using various log files, you can, for example, check and compare the results of touch-probe cycles.

  The control needs other function-dependent log files to, for example, detect process variations.

For more details on data acquisition, please refer to the sections describing the individual functions and the **Settings** application.

Data transfer and data access

Machine manufacturers, machine operators, or users can transfer data to external data carriers or drives, for example, and set up and configure a data transfer functionality. The user has control over and responsibility for the stored data (e.g., with respect to safety and the duration of storage).

Examples of data transfer:

- **HEIDENHAIN DNC** (#18 / #3-03-1)

  This software option enables external Windows applications to access data of the control via the TCP/IP protocol.
- **OPC UA NC Server** (#56-61 / #3-02-1*)

  These software options include OPC UA, a standardized interface that allows external access to the data and functions of your control.
- USB devices

  USB devices can be used to transfer data or make external backups.

> For more details on data transfer, please refer to the sections describing the individual functions and the **Settings** application.

Data access via file management:

- Drives, folders, and files

  You can access external data or make external backups by using connected drives or USB devices. For example, you can manage, create, edit, or delete folders and files.
- Functions and settings

  In the file manager you can search for certain contents or display hidden or dependent files.

> For further details on file access, please refer to the section on the **Files** operating mode.
>
> Please observe the safety instructions and information given by the machine manufacturer in order to avoid loss of data or jeopardize the functionality of your system.

3.1.3 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 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 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 73

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
817620-20	TNC7
817621-20	TNC7 E
817625-20	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. The machine manufacturer can also change the colors in the control's GUI.

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. The export version is not covered by Annex I of the EU Dual-Use Regulation. In this version, the Adv. Function Set 2 (#9 / #4-01-1) software option 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 587

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, for example: (#18 / #3-03-1).
>
> The Technical Manual contains information 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 586

Overview

Keep in mind that particular software options also require hardware extensions.

Further information: "Hardware", Page 82

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 an unrolled 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.
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 362
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 291
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 280
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

Software option	Definition and application
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 611
4 Additional Axes (#77 / #6-01-1*)	**Four additional control loops** **Further information:** "**Control Loop Qty.** (#0-7 / #6-01-1*)", Page 74
8 Additional Axes (#78 / #6-01-1*)	**Eight additional control loops** **Further information:** "**Control Loop Qty.** (#0-7 / #6-01-1*)", Page 74
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

Software option	Definition and application
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 518 ■ 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 516 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 488 ■ **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 don't need to move between different workstations as much, and so increase your efficiency. **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 628

Software option	Definition and application
Collision Monitoring v2 (#140 / #5-03-2)	**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 ■ Collision monitoring of fixtures **Further information:** "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258 ■ Defining a reduced minimum distance between fixture and tool With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or machine downtime.
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 290
Machine Vibr. Contr. (#146 / #2-24-1)	**Vibration damping for machines (MVC)** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**

Software option	Definition and application
CAD Model Optimizer (#152 / #1-04-1)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 370
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

Software option	Definition and application
Turning v2 (#158 / #4-03-2)	**Mill-turning version 2** This software option includes all functions of the Turning software option (#50 / #4-03-1). In addition, this software option offers the following advanced turning functions: ■ Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** ■ Cycle **883 TURNING SIMULTANEOUS FINISHING** The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation. **Further information:** User's Manual for Machining Cycles
Model Aided Setup (#159 / #1-07-1)	**Graphically supported setup** This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model. **Further information:** "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 411
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 303

Software option	Definition and application
PTI 8 channels (#3-04-1*)	**Process Tracking Interface PTI** This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with **SIK2**. You can order this software option up to two times, each time selecting up to eight channels for recording. **Further information:** "Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)", Page 714 For more information, refer to the integrated help system of RemoTools SDK. ID 340442-xx

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 License 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 611

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 72

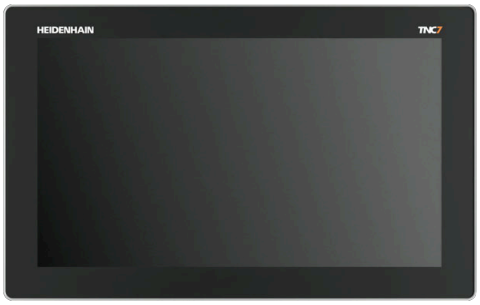
The actual range of functions also depends on hardware enhancements and the enabled software options.

Further information: "Hardware enhancements", Page 86

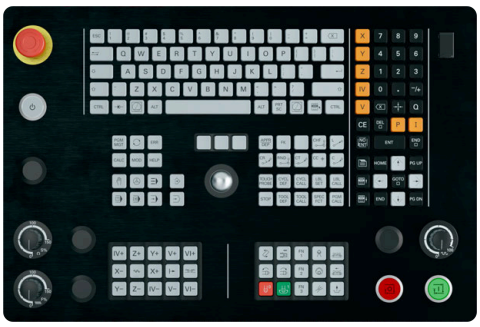
Further information: "Software options", Page 73

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 97

Further information: "Operating elements of the keyboard unit", Page 97

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.

To navigate to this function:

Settings application ▶ **Diagnostics/Maintenance group** ▶ **Touchscreen Cleaning**

Further information: "The Settings application", Page 575

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.

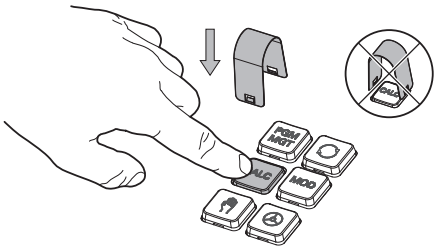
- ▶ Apply a small amount of the cleaning agent onto a cleaning cloth
- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

Exchanging keycaps

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

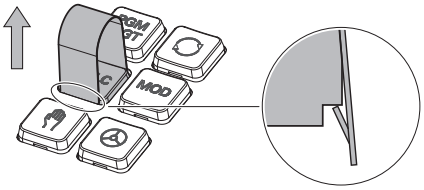
IP54 protection cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

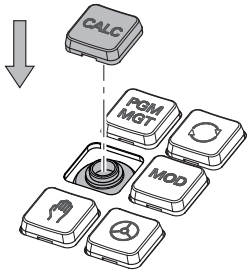


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

  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.

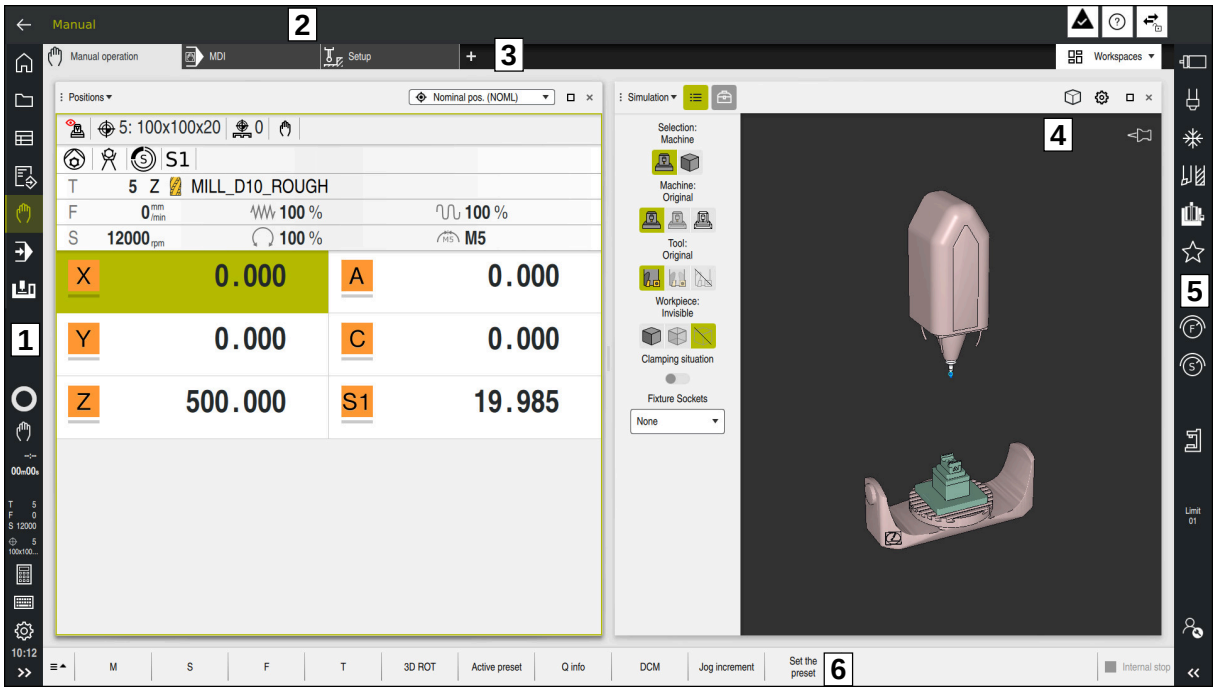
Keep in mind that particular hardware enhancements require additional software options.

Further information: "Software options", Page 73

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 535
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 387
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 628 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 553

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 90
- Status overview

  Further information: "Status overview of the TNC bar", Page 133
- Calculator

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

  Further information: "Virtual keyboard of the control bar", Page 376
- 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.

    This manual describes the state of the GUI with inactive **Dark Mode**. Your control might display other colors than shown in this User's Manual.
  - **Font size**

- Date and time

2 Information bar
- Active operating mode
- Message menu

  Further information: "Message menu on the information bar", Page 379
- Symbols

  Further information: "Icons on the control's user interface", Page 105

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 92

5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.

6 Function bar
- Selection menu for buttons

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

  With the buttons you activate individual functions of the control.

Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:
- Operating modes
- Applications
- Workspaces
- Buttons

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 575 See the User's Manual for Programming and Testing Page 646
	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 456
	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 170 Page 383 Page 387 Page 165 Page 452
	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 426
	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 563
	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.	

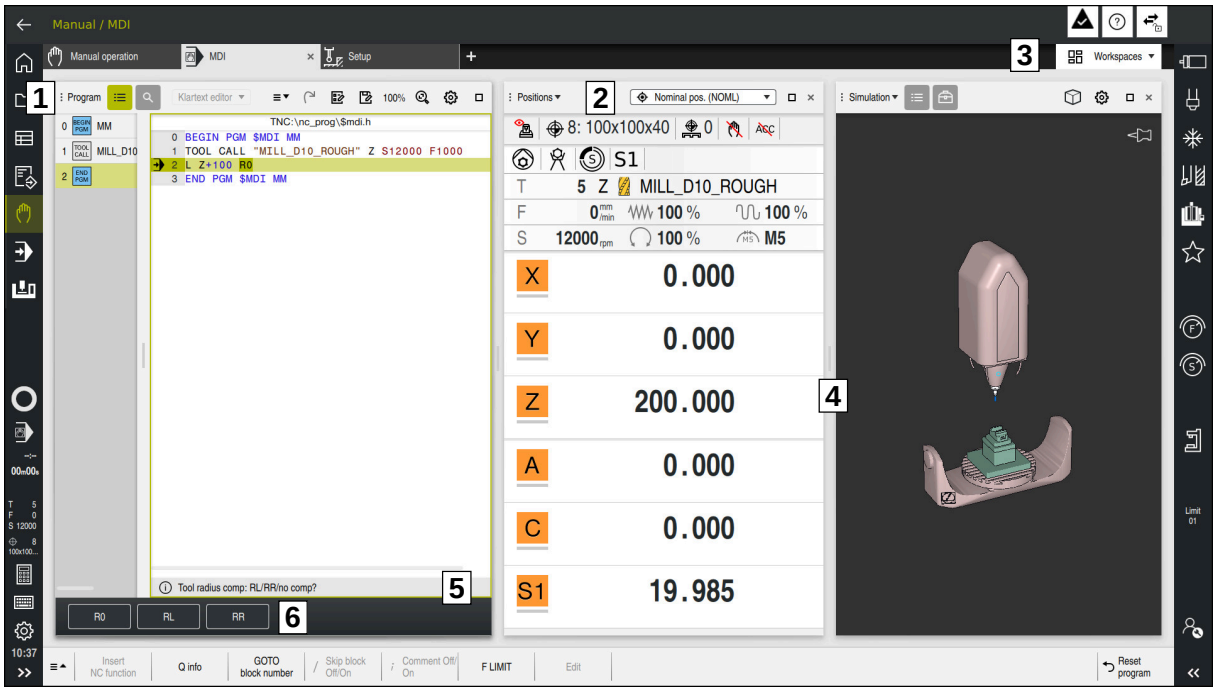
Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to gray out or hide the following user interface elements, for example:

- Operating modes
- Applications
- Buttons

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 The **Move** icon

Use the **Move** icon in the title bar to change the position of the workspaces. You can also align two workspaces vertically above each other.

Alternatively, you can drag the title to move a workspace.

2 Title bar

In the title bar the control shows the title of the workspace, and different icons or settings, depending on the workspace.

When you select the title of a workspace, the control will open a selection menu. This selection menu allows you to switch workspaces without changing their size or position.

You can only switch between workspaces that can be closed.

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 Dialog bar

In the dialog bar, the control shows information on the current input, such as a description or unit of measure.

6 Action bar

In the action bar the control shows selection options for the current dialog; for example, an NC function.

3.7.2 Icons within the workspaces

In the title bar, the control displays the following icons:

Icon	Meaning
▼	Open the **Workspaces** selection menu Only for workspaces that can be closed
☐	**Maximize** Only if at least two workspaces are open
⧉	**Reduce** Only if at least two workspaces are open
✕	**Close** Only if at least two workspaces are open that can be closed

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.

Icons that are specific to individual workspaces are described there.

An overview of the icons that are accessible from all operating modes or can be used in multiple operating modes is found here:

Further information: "Overview of icons not specific to any operating mode", Page 105

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 387
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 manager, 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 651
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

Workspace	Further information
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 575
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 470
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
Retract In the **Retract** workspace, you can disengage the tool after a power interruption.	Page 452
GPS (#44 / #1-06-1) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	Page 291
Desktop menu In the **Desktop menu** workspace, the control displays selected control and HEROS functions.	Page 108
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 647
Positions In the **Positions** workspace, the control displays information about the status of various functions of the control and about current axis positions.	Page 127
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 319

Workspace	Further information
Referencing On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.	Page 165
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 563
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
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 155
Start In the **Start** workspace, the control shows the steps that are performed during startup.	Page 113
Status In the **Status** workspace, the control shows the status and values of individual functions.	Page 135
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 462
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 647
Keyboard In the **Keyboard** workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 376
Text editor Use the **Text editor** workspace to create and edit text files.	See the User's Manual for Programming and Testing
Overview In the **Overview** workspace, the control displays information on the status of individual functional safety (FS) safety functions.	Page 570

Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:

- Operating modes
- Applications
- Workspaces
- Buttons

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 97

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 376

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 the window or context menu, discard the current input
⇄	Select the next element; for example, an input field, button, or selection option
[SHIFT] + [TAB]	Select the previous element
⌫	Show the hidden NC block
? /	Hide or show 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 630
	Open context menu
+ = – * 8 ? / (9) 0	Perform 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 379
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 575
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 58

Operating modes

On the TNC7, the operating modes of the control are allocated differently than on the previous controls. 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 170
	Activate and deactivate the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic handwheel", Page 535
	Open the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 203
	Open the **MDI** application in the **Manual** operating mode **Further information:** "The MDI application ", Page 383
	Open the **Program Run** operating mode in **Single Block** mode **Further information:** "The Program Run operating mode", Page 426
	Open the **Program Run** operating mode **Further information:** "The Program Run operating mode", Page 426
	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.

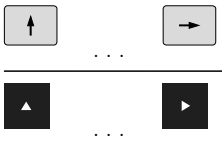
Key	Meaning
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function
FK	Open the **Contour graphics** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode
CHF	Program a chamfer
L	Program a straight 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 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., to program 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

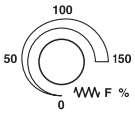
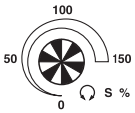
Keycaps for axis input and value input

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 of the TNC bar", Page 133 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
	■ Directly position the cursor by using the number of an NC block, table row or machine parameter ■ Open the selection menu while editing
	Jump to first line of an NC program or first column of a table
	Jump to last line of an NC program or last column of a table
	Go one page up in an NC program or table
	Go one page down in an NC program or table
	Mark the active application in order to navigate between applications
	Navigate between areas of an application

Potentiometers

Potentiometer	Function
	Increase or reduce the feed rate **Further information:** 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**
	**Keyboard** for formula input
	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**
	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.
	Activate or deactivate **Change column width** You can change the width of the currently selected column.

Icon or shortcut	Meaning
	Scale Resize windows
	File functions are available
	■ Black: **Add favorite** ■ Yellow: **Remove favorite**
	Edit
 **[CTRL] + [S]**	**Save**
	Save as
 **[CTRL] + [F]**	**Find**
 **[CTRL] + [X]**	**Cut**
 **[CTRL] + [C]**	**Copy**
 **[CTRL] + [V]**	**Paste**
 **[CTRL] + [Z]**	**Undo**
 **[CTRL] + [Y]**	**Redo**
100%	Font size in the workspace If you select the percent value, the control displays icons for increasing and decreasing the font size.
 **[CTRL] + [-]**	**Reduce**
 **[CTRL] + [+]**	**Magnify**
	Set the workspace font size to 100%
 	Open or close the selection menu Depending on the size of the workspace, the control gathers, for example, the icons in the title bar into a selection menu.
	Open or close the **Workspaces** selection menu

Icon or shortcut	Meaning
	Show the **Message menu**
	Call context-sensitive help **Further information:** "Context-sensitive help", Page 61
	Dynamic Collision Monitoring DCM (#140 / #5-03-2) deactivated **Further information:** "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
	**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 447
	**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 627 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
[CTRL] + [A]	Mark everything

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 653
- Full-text search

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

  Further information: "Adding and removing favorites", Page 109

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 90

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

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

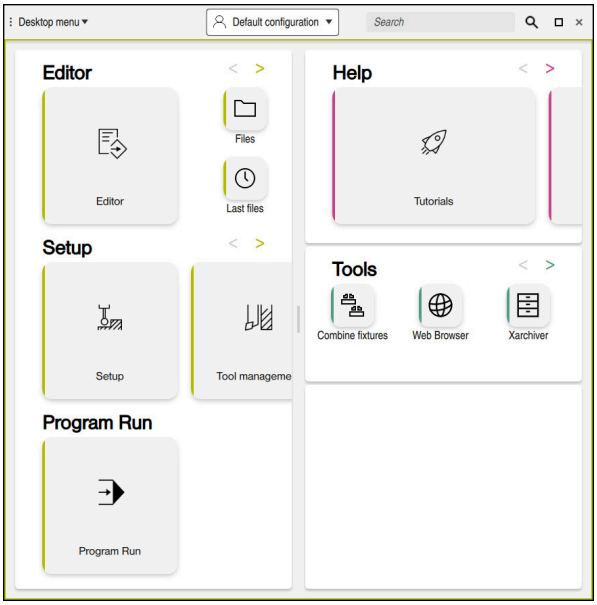
  Further information: "HEROS operating system", Page 683
- **Help**

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

  Further information: "User's Manual as integrated product aid: TNCguide", Page 58
- **Favorites**

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

  Further information: "Adding and removing favorites", Page 109



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:

- ▶ Enter the search term in the full-text search (e.g., **dat**)
- ▶ Long-press or right-click the function's icon (e.g., **Date/Time**)
- > 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 the machine and the control on
- Setting up a tool
- Setting up a workpiece
- Machining a workpiece
- Switching the machine off

4.2 Switching the machine and the control on

The **Start** 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 on the power supply of the control and of the machine

> The control is in start-up mode and shows the progress in the **Start** workspace.

> The control displays the **Power interrupted** dialog in the **Start** workspace.

OK

- Select **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 165

- ▶ 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 170

More detailed information

- Switching on and off
- Position encoders

  Further information: "Position encoders and reference marks", Page 181
- Axis reference run

4.3 Setting up a tool

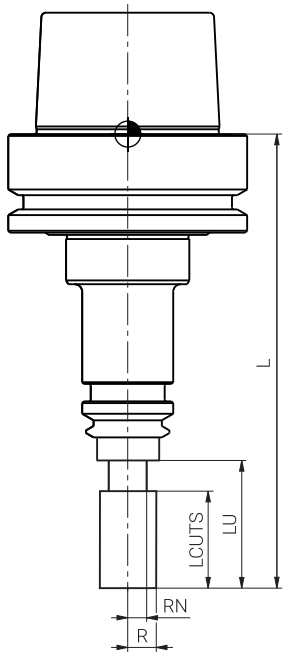
This chapter uses an example to show how to set up a tool.

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

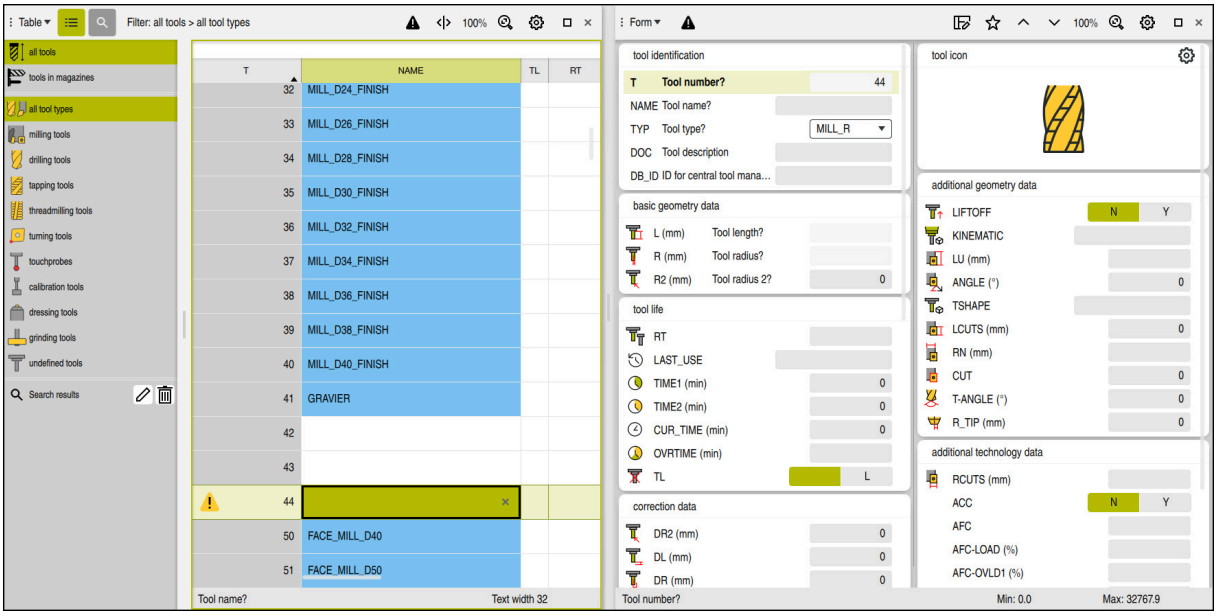
To prepare tools:

- ▶ 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 the tool by scratching", Page 419
- 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 the **Tool management** application
- > The control displays the **Tool management** application.
- ▶ In the application bar, select **Workspaces**
- ▶ Select **Form**
- > The control opens the **Form** workspace.

More detailed information

- **Tables** operating mode
  Further information: "The Tables operating mode", Page 456
- The **Form** workspace
  Further information: "The Form workspace for tables", Page 470
- Tool parameters
  Further information: "Tool parameters", Page 191
- Tool management
  Further information: "Tool management ", Page 203

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 200

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 **CUT** (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.

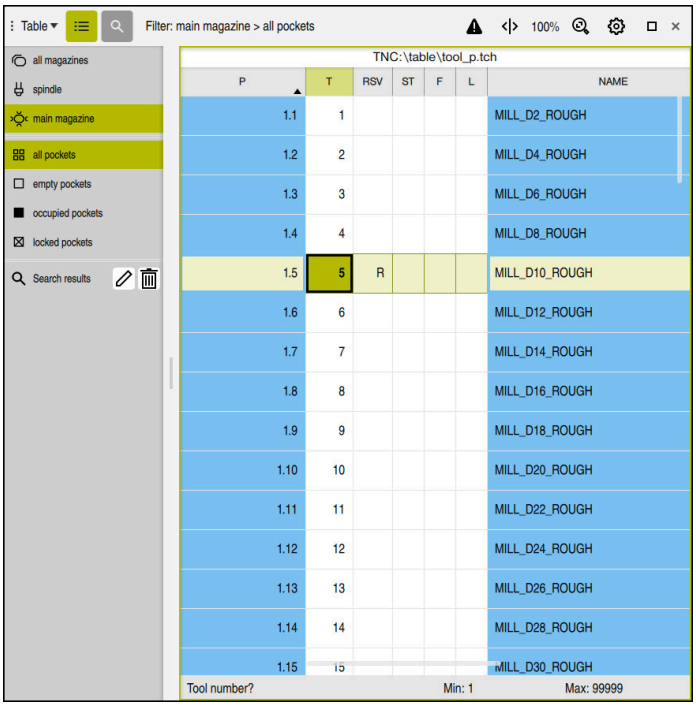
Further information: Programming and Testing User's Manual

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 191

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.



The **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 511

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 90

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 387

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+**)

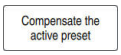
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the first workpiece edge

- 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+**)

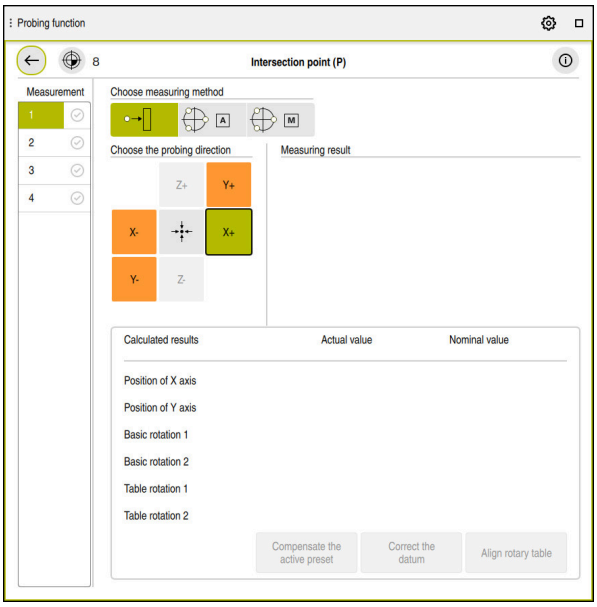
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the second workpiece edge

- 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 387
- Presets in the machine

  Further information: "Presets in the machine", Page 182
- Preset management

  Further information: "Preset management", Page 233
- Tool change in the **Manual operation** application

  Further information: "The Manual operation application", Page 170

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

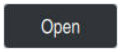
- The **Program Run** operating mode
  Further information: "The Program Run operating mode", Page 426

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.

4.5.4 Interrupting an NC program

To interrupt an NC program:

- Press the **NC stop** key
- > The control will interrupt the active machining operation.

While a machining operation is interrupted, it is not possible to change the operating mode and edit the NC program.
To continue program run, press the **NC Start** key.

4.5.5 Aborting program run

After program run has been aborted, it is possible to change the operating mode and edit the NC program.

To abort program run:

- Press the **NC stop** key
> The control will interrupt the active machining operation.

- Select **Internal stop**
> The control aborts program run.

> For a safe restart of the NC program, always use a block scan. This ensures that the control will take the entire program contents into account.

4.5.6 Safe restart with block scan

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. In a block scan, the control restores the machine status up to the corresponding NC block (i.e., the spindle is switched on or the machine axes are tilted to position). A block scan reduces the risk of collision.

Running a block scan after a program interruption

To perform a block scan:

- Select the **Program Run** operating mode

- Select **Block scan**
> The control opens the **Block scan** window, displaying the data of the last executed NC block.
- If required, enter the **Block number** (e.g., **29**)

- Press the **NC Start** key
> The control starts the block scan and calculates until the entered NC block.
> The control opens the **Restore machine status** window.

- Press the **NC Start** key
> The control restores the machine status up to the defined NC block (e.g., **TOOL CALL** or M functions).
> The control displays the **Axis sequence for return to contour:** window. The **Axis sequence for return to contour:** window contains the positioning logic required to reach the position before the defined NC block.

- Press the **NC Start** key
> Using the displayed positioning logic, the control moves to the required position.

- Press the **NC Start** key
> The control resumes execution of the 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 offers the following status displays:

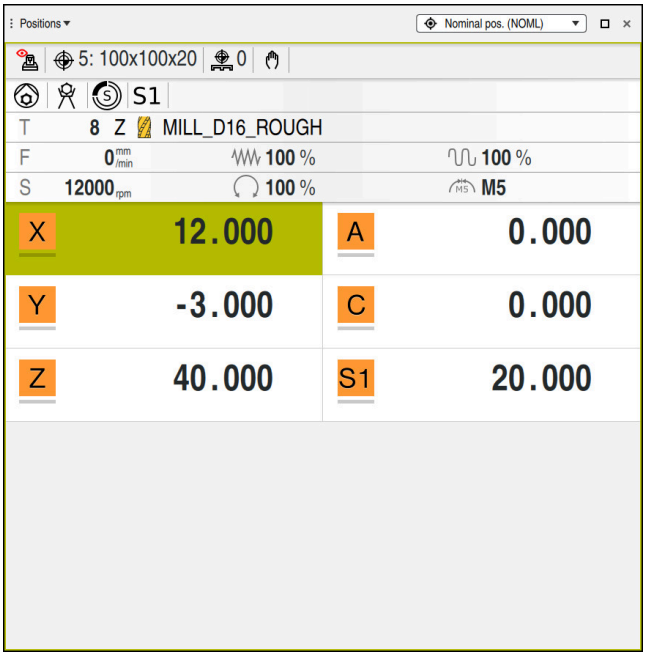
- General status display and position display in the **Positions** workspace
  Further information: "The Positions workspace", Page 127
- Status overview on the TNC bar
  Further information: "Status overview of the TNC bar", Page 133
- Additional status displays for specific areas in the **Status** workspace
  Further information: "The Status workspace", Page 135
- 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 155

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 90

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

- Icons of active and inactive functions (e.g., Dynamic Collision Monitoring (DCM) (#140 / #5-03-2)
- 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 572

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 158
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 132
?	The axis is not referenced.
⚠	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 573
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 570
S_{SOS} 0 U/min	Spindle status when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 570

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 233
	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 103 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 431
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 426
	Tool radius compensation **RL** is active. **Further information:** Programming and Testing User's Manual
	Tool radius compensation **RR** is active. **Further information:** Programming and Testing User's Manual The control displays these symbols as transparent during the **Block scan** function. **Further information:** "Block scan for mid-program startup", Page 438
	Tool radius compensation **R+** is active. **Further information:** Programming and Testing User's Manual
	Tool radius compensation **R-** 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 438
	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 438
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 235
	The basic rotation will be taken into account while moving the axes. **Further information:** "The Basic rotation selection item", Page 245
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 235

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 245
	The **Tool axis** function is active. **Further information:** "The Tool axis selection item", Page 245
	The **TRANS MIRROR** function or the Cycle **8 MIRRORING** is active. The axes programmed in this function or cycle are moved with mirroring. **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
	The **FUNCTION DRESS** dressing mode is active (#156 / #4-04-1). **Further information:** Programming and Testing User's Manual

Icon	Meaning
	The Dynamic Collision Monitoring function (DCM (#140 / #5-03-2)) is active.
	The Dynamic Collision Monitoring function (DCM (#140 / #5-03-2)) is not active. **Further information:** "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
	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 280
ACC	The Active Chatter Control function (ACC) is active (#145 / #2-30-1). **Further information:** "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 290
	The Global program settings function (GPS) function is active (#44 / #1-06-1). **Further information:** "Global Program Settings (GPS) (#44 / #1-06-1)", Page 291
	The Process monitoring function is active (#168 / #5-01-1). **Further information:** "Process monitoring (#168 / #5-01-1)", Page 303

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 (#140 / #5-03-2)) 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.

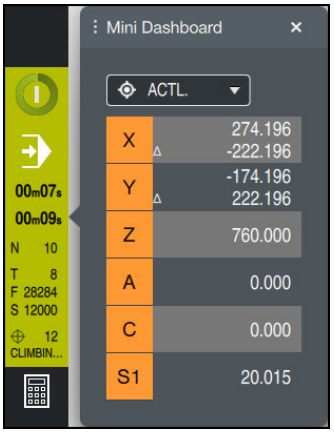
5.3 Status overview of the TNC bar

Application

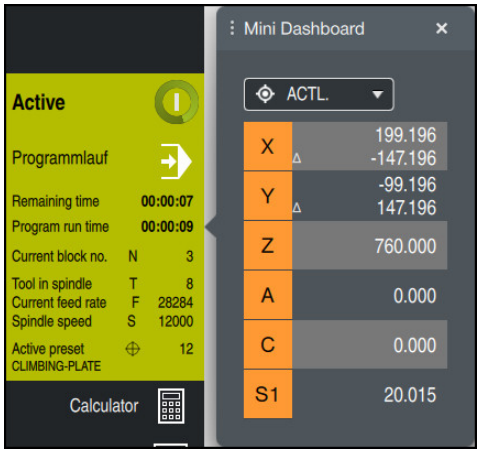
On the TNC bar, the control shows a status overview with the execution status, the current technology values, and the axis positions.

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 134
- 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 156

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 127

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 432

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 90

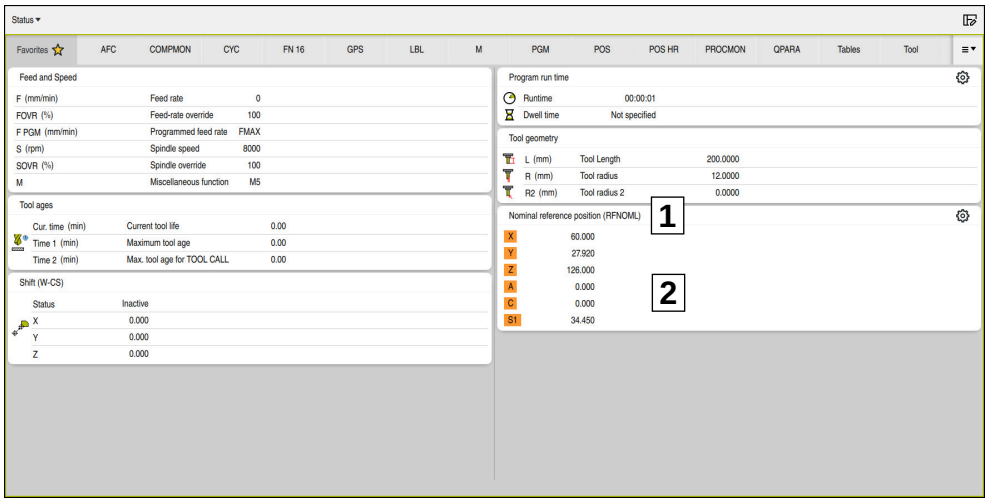
Icons

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 136
	**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 472 ■ 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 the contents of the other tabs.



The **Favorites** tab

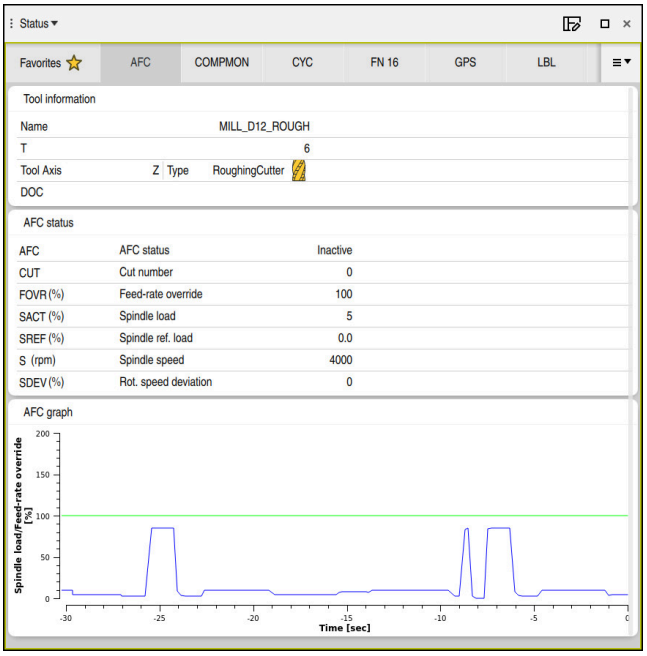
1 Area
2 Contents

Each section 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 280



AFC tab

Area	Contents
Tool information	■ **Name** Tool name ■ **T** Tool number ■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**) ■ **Doc** Comment about the tool from the tool management

Area	Contents
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 283 ■ **S (rpm)** Spindle shaft speed in rpm ■ **SDEV (%)** Current deviation of the speed in percent
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.

- By scrolling or dragging the graph, you can enlarge or reduce its size horizontally.
- You can shift the graph by swiping or by dragging 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.

Area	Contents
Enforced reactions of component monitoring	Table with all the reactions triggered since the time the control was restarted The control displays the following information for each reaction: ■ Reaction icon ■ Timestamp ■ Monitoring task ■ Path of the NC program ■ Tool position When you select a cell in the header row, the control sorts the table accordingly in ascending or descending order. Selecting the **Filters** icon allows you to enter a search term for each column and to filter by it. The control displays only rows that contain the search term. You can select whether the control should take case sensitivity into account. Reselecting the **Filters** icon deactivates the filter. You can delete all of the reactions via a button.

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** (#140 / #5-03-2) If a value is limited by DCM, then the control displays the text **DCM limited** following the value.

The machine manufacturer defines the tolerance limits using Dynamic Collision Monitoring (DCM (#140 / #5-03-2)).

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 Cycle **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 291

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 223
Shift (W-CS)	■ **Status** ■ **X** **Shift (W-CS)** in the X axis The **Shift (W-CS)** function is also available for the other linear axes **Y** and **Z**.
Mirroring (W-CS)	■ **Status** ■ **X** **Mirroring (W-CS)** in the X axis The **Mirroring (W-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.
Rotation (WPL-CS)	■ **Status** ■ **(°)** **Rotation (WPL-CS)** in degrees The **Rotation (WPL-CS)** function is active in the working plane coordinate system **WPL-CS**. Entries are in degrees. **Further information:** "Working plane coordinate system WPL-CS", Page 226
Shift (mW-CS)	■ **Status** ■ **X** **Shift (mW-CS)** in the X axis The **Shift (mW-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.

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

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 156
Programs called	Path of the main program as well as called NC programs including the path
Pole/circle center	Programmed axes and values of the circle center point **CC**
Radius compensation	Programmed tool radius compensation
Program run options	Active breakpoints in connection with the override controller **Further information:** "Override controller", Page 553

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)** ■ **Actual distance to go (ACTDST)** ■ **Nominal distance to go (REFDST)** **Further information:** "Position displays", Page 158

Area	Contents
Feed and Speed	■ Active **Feed** in mm/min Only in the **Status** workspace 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 431 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 569 ■ Active **Feed-rate override** in % Only in the **Status** workspace ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min Only in the **Status** workspace 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 % Only in the **Status** workspace ■ 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.
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 245
OEM transformation	The machine manufacturer can define an OEM transformation for special turning kinematics. **Further information:** "Definitions", Page 154

Area	Contents
Basic transformations	The control displays the values of the current workpiece preset in this area. **Further information:** "Preset management", Page 233
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 291
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 303

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
Process monitoring enforced reactions	Table with all the reactions triggered since the time the control was restarted The control displays the following information for each reaction: ■ Symbol of the reaction type (e.g., a warning) ■ Timestamp ■ Monitoring task ■ Path of the NC program or the called program ■ Monitoring section ■ Tool number ■ Tool position When you select a cell in the header row, the control sorts the table accordingly in ascending or descending order. Selecting the **Filters** icon allows you to enter a search term for each column and to filter by it. The control displays only rows that contain the search term. You can select whether the control should take case sensitivity into account. Reselecting the **Filters** icon deactivates the filter. You can delete all of the reactions via a button.

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 160

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
Compensation value 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 86

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 154 ■ **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 200

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**) ■ **TO** Tool orientation For all tools except for milling and drilling tools ■ **DOC** Note on the tool For all tools except for dressing tools (#156 / #4-04-1)
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 203
Tool ages	■ **Cur. time (min)** Current amount of time during which the tool has been engaged ■ **Time 1 (min)** Service life of the tool ■ **Time 2 (min)** Maximum service life at tool call

Area	Contents
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool

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)** Tool tip 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 226 ■ **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 226

Deviating contents for grinding tools (#156 / #4-04-1)

Area	Contents
Tool geometry	■ **LO (mm)** Total length in Z direction ■ **R-OVR (mm)** Radius of replacement tool ■ **B (mm)** Width of grinding wheel ■ **RV (mm)** Radius at the edge for L-OVR ■ **RV1 (mm)** Radius at the edge for LO
Tool allowances	■ **dR-OVR (mm)** Delta value of radius ■ **dLO (mm)** Delta value of total length

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 136
- **CYC**
  Further information: "CYC tab", Page 141
- **FN 16**
  Further information: "The FN 16 tab", Page 141
- **LBL**
  Further information: "The LBL tab", Page 143
- **M**
  Further information: "The M tab", Page 143
- **PGM**
  Further information: "The PGM tab", Page 144
- **POS**
  Further information: "The POS tab", Page 145
- **QPARA**
  Further information: "The QPARA tab", Page 149
- **Tables**
  Further information: "The Tables tab", Page 149
- **TRANS**
  Further information: "The TRANS tab", Page 149
- **TT**
  Further information: "The TT tab", Page 151
- **Tool**
  Further information: "The Tool tab", Page 152

Notes

- The control displays the current counter reading and the defined target quantity only on the **PGM** tab of the **Status** workspace.
  Further information: "The PGM tab", Page 144
  Further information: Programming and Testing User's Manual
- The following contents are only displayed on the **POS** tab of the **Status** workspace:
  - Active **Feed** in mm/min
  - Active **Feed-rate override** in %
  - Active **Spindle override** in %

  Further information: "The POS tab", Page 145

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 144
- Status overview on the TNC bar
  Further information: "Status overview of the TNC bar", Page 133
- **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 212

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 432

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

Mode	Meaning
Actual distance to go (ACTDST)	This mode displays for every axis the difference between the actual position and target position in the **I-CS** input coordinate system. The control takes all the coordinate transformations into account and displays the remaining distance to the programmed target position. Example: ■ **TRANS ROTATION ROT+45** is active ■ The **L IX+10** positioning block has been programmed The status display shows a distance-to-go only in **X** because rotation is taken into account.
Nominal distance to go (REFDST)	This mode displays for every axis the difference between the actual position and the target position in the machine coordinate system **M-CS**. The control displays the distance that every physical axis actually moves and does this independently of the active coordinate transformations. Example: ■ **TRANS ROTATION ROT+45** is active ■ The **L IX+10** positioning block has been programmed The status display shows the distances-to-go in **X** and **Y** because both axes are moved.

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.
- If the axes are moved, then, in addition to the current positions, the control also displays the remaining distances-to-go with a symbol and the respective value.

  Exception: The **Actual distance to go (ACTDST)** and **Nominal distance to go (REFDST)** modes

  Further information: "Axis display and position display", Page 128

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 149

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 Switch-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 181

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 control 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 165
  Further information: "The Manual operation application", Page 170

6.1.1 Powering the machine and the control on

To switch the machine on:

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

- ▶ Select **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 165

- ▶ 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 170

If startup is delayed due to 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 570

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

- Once startup has been completed, the control will close the **Start** workspace.

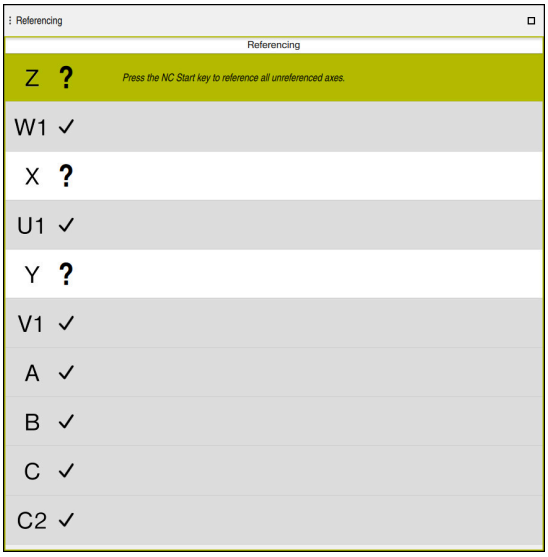
6.2 The Referencing workspace

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 628

7

Manual operation

7.1 The Manual operation application

Application

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

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 172
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 173

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
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 520
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
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 175 ■ **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 176
DCM	The control opens the **Dyna. Coll. Monitoring (DCM)** window where you can activate or deactivate Dynamic Collision Monitoring (DCM (#140 / #5-03-2)). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 252
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 173
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 233
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 203
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic handwheel", Page 535
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 432

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 387

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 383

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 535

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

Further information: "Status displays", Page 125

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 of the TNC bar", Page 133

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**)

X+

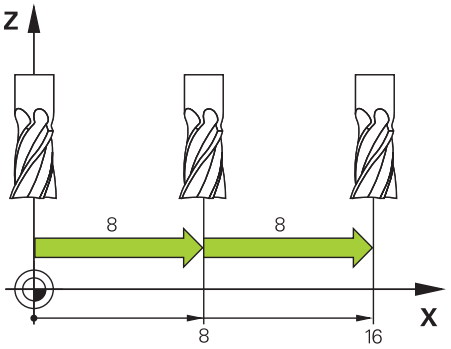
- ▶ Press the axis key of the desired axis
- > The control moves the axis as long as you press the key.

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

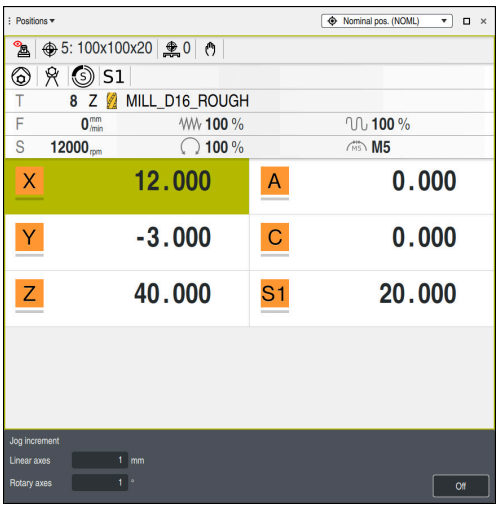
Incremental jog positioning allows you to move a machine axis by a preset distance. The input range for the infeed is from 0.001 mm to 10 mm.



To position an axis incrementally:

- Select the **Manual** operating mode
- Select the **Manual operation** application
- Jog increment
  - Select **Jog increment**
  - > The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
  - Enter the jog increment for linear axes and rotary axes
- X+
  - Press the axis key of the desired axis
  - > The control positions the axis in the selected direction by the defined jog increment.
- Off
  - 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 175
Measure unbalance	Detect the unbalance of current clamping for turning and calculate suggestions for balance weights	Page 176

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 176
- 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 [mm]** or **Radial position [inches]** Distance from the table center ■ **Weight [g]:** or **Mass [lb]**
Alternative settings	■ **Weight [g]:** or **Mass [lb]** ■ **Radial position [mm]** or **Radial position [inches]** 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. The angle is equivalent to the angle from the **Proposed unbalance** area. 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

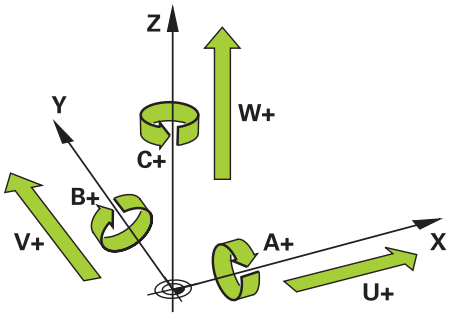
To compensate for an unbalance, several balancing weights at different positions may be required.

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:

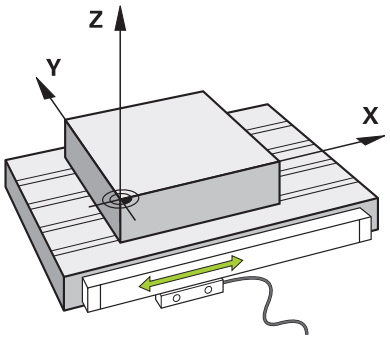
Main axis	Secondary axis	Tool axis	Working plane
X	**Y**	**Z**	**XY**, also **UV**, **XV**, **UY**
Y	**Z**	**X**	**YZ**, also **WU**, **ZU**, **WX**
Z	**X**	**Y**	**ZX**, also **VW**, **YW**, **VZ**

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 216

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

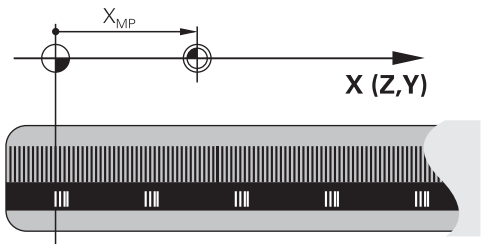
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

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 165



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 185

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 218 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 181 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 165
	**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 223 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 233 If no transformations are defined, the entries in the NC program refer to the workpiece preset.
	**Workpiece datum** You define the workpiece datum with transformations in the NC program, for example with **TRANS DATUM** or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset. If you tilt the working plane (#8 / #1-01-1), the workpiece datum is the point around which the workpiece is rotated.

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 185
- 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 191
- Save all collected parameters of this tool in the tool management.

  Further information: "Tool management ", Page 203
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 205
- 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 212
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: Programming and Testing User's Manual

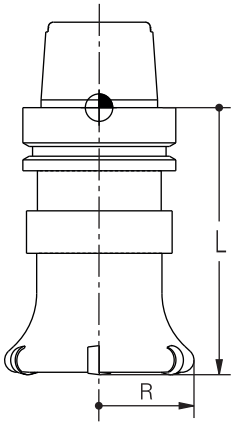
9.2 Presets on the tool

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

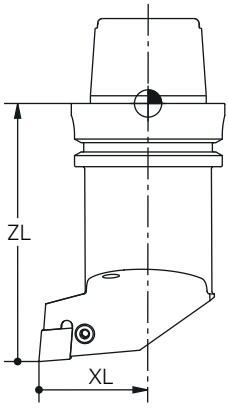
Related topics

- Presets in the machine or on the workpiece
  Further information: "Presets in the machine", Page 182

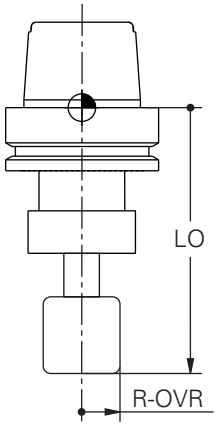
9.2.1 Tool carrier reference point



Milling cutter



Turning tool



Grinding tool

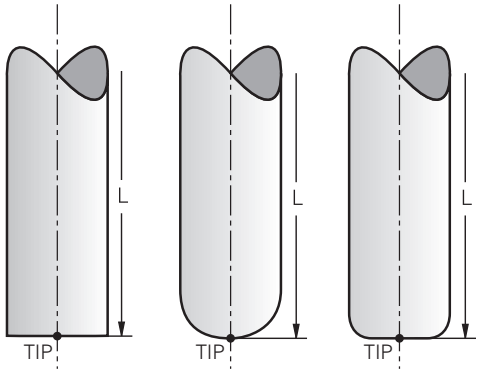
The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

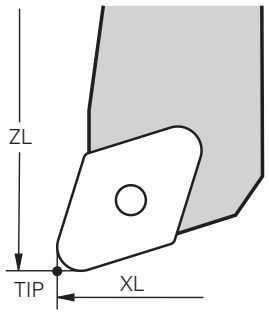
Further information: "Tool management ", Page 203

Further information: "Measuring the tool by scratching", Page 419

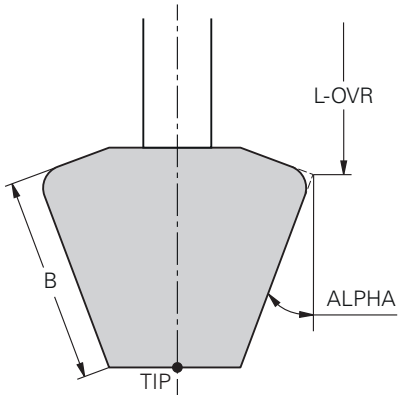
9.2.2 Tool tip TIP



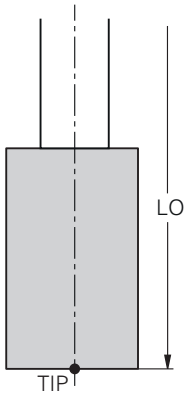
Milling tools



Turning tools



Grinding tools



Further information: "Tool coordinate system T-CS", Page 230

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 191

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

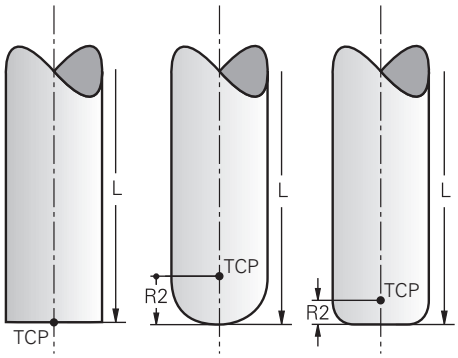
For the following grinding tools, the control calculates the longest point of the tool based on multiple parameters:

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

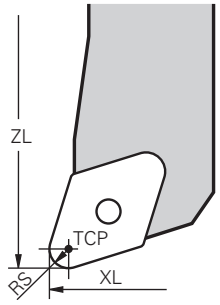
The tool tip is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 188

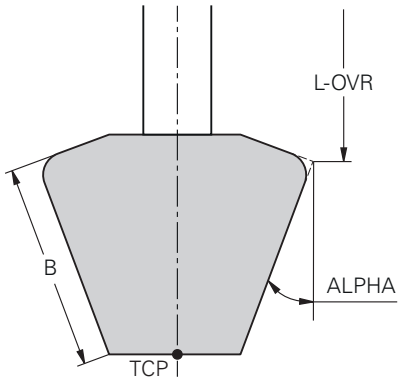
9.2.3 Tool center point (TCP, tool center point)



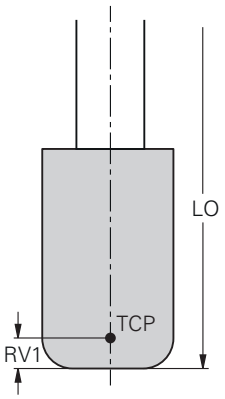
Milling tools



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.

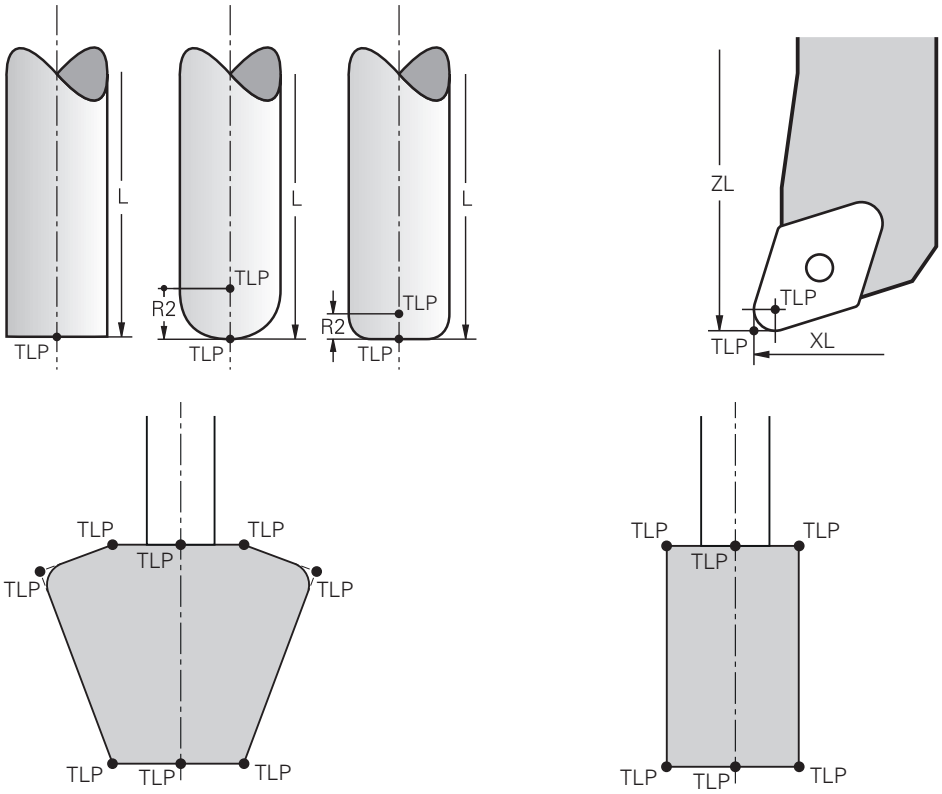
Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool management ", Page 203

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 188

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.

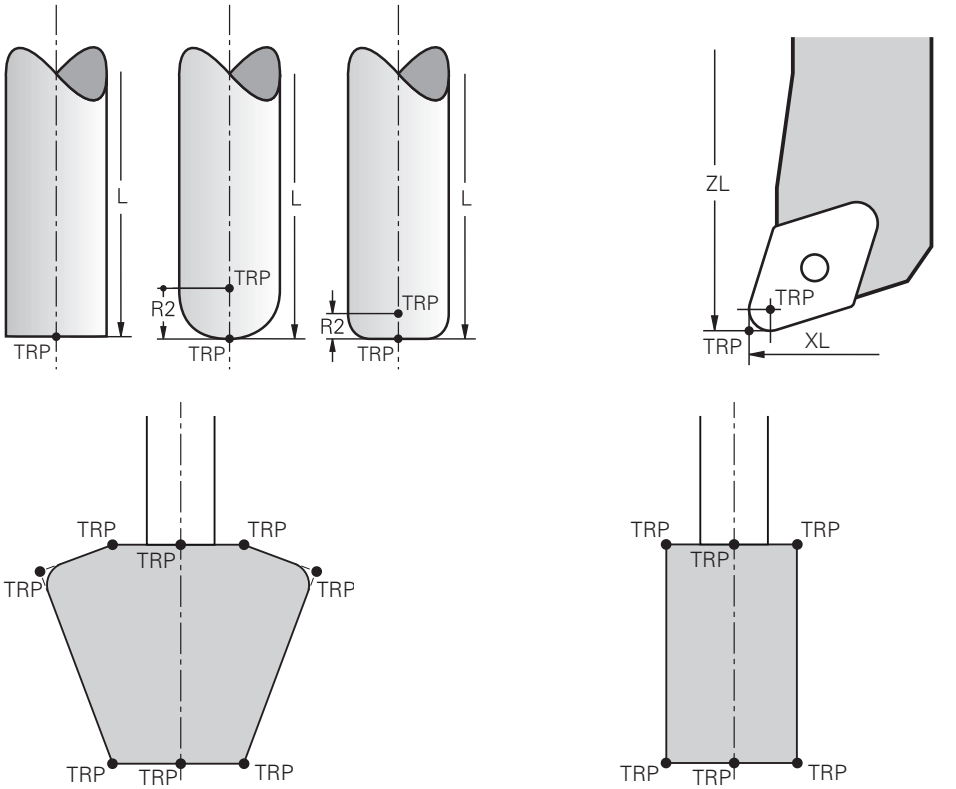
In the function **FUNCTION TCPM** (#9 / #4-01-1), you can also choose the tool location point to be at the tool center point.

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.

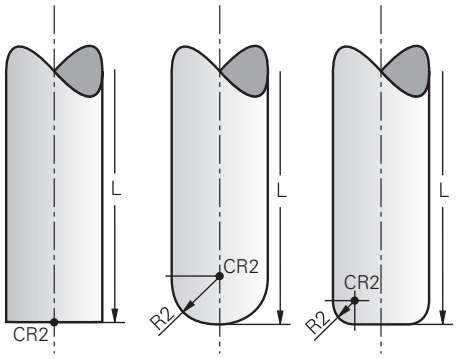
Further information: Programming and Testing User's Manual

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 188

9.3 Tool parameters

Application

In the tool parameters you give the control all the information it needs 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 203
- Tool types
  Further information: "Tool types", Page 200
- Tool tables
  Further information: "Tool tables", Page 478
- Tool table **tool.t**
  Further information: "Parameters of the tool table tool.t", Page 479
- Turning tool table **toolturn.trn** (#50 / #4-03-1)
  Further information: "Parameters of the turning tool table toolturn.trn", Page 489
- Grinding tool table **toolgrind.grd** (#156 / #4-04-1)
  Further information: "Parameters of the grinding tool table toolgrind.grd", Page 495
- Dressing tool table **tooldress.drs** (#156 / #4-04-1)
  Further information: "Parameters of the dressing tool table tooldress.drs", Page 504
- Touch probe table **tchprobe.tp**
  Further information: "Parameters of the touch probe table tchprobe.tp", Page 507

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 when you enter the parameters. In the fillable form, the control filters the parameters in accordance with the selected tool types.

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: User's Manual for Machining Cycles
- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
  Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248

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 203

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 replaces 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 280

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 478

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 194

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 185

Further information: "Creating an indexed tool", Page 195

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 the **Tool management** application

- 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 191

- 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 191

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 185

Notes

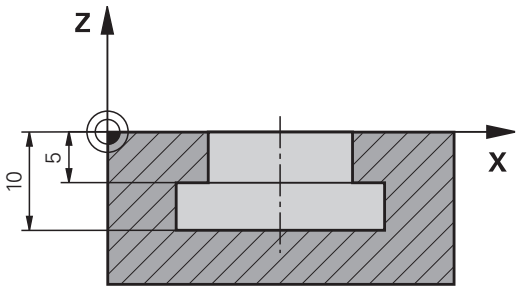
- The control describes some parameters automatically, for example the current tool age **CUR_TIME**. The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 478
- 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.

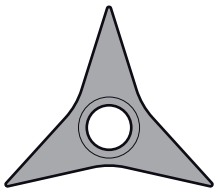
The T-slot is programmed in two machining steps:

- 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

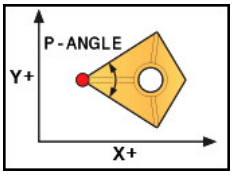
Example: FreeTurn tool (#50 / #4-03-1)

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 185
 **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 185
 **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 201
 **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 203
- Tool tables
  Further information: "Tool tables", Page 478

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
	**CAL_PIN**	28
	Turning tool (**TURN**) (#50 / #4-03-1) **Further information:** "Turning tool types (#50 / #4-03-1)", Page 201	29
	Touch probe (**TCHP**)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5

Icon	Tool type	Number
	Piloted counterbore (**TSINK**)	6
	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 milling drill (**ZBGF**)	20
	Grinding wheel (**GRIND**) (#156 / #4-04-1) **Further information:** "Grinding tool types (#156 / #4-04-1)", Page 202	30
	Dressing tool (**DRESS**) (#156 / #4-04-1) **Further information:** "Dressing tool types (#156 / #4-04-1)", Page 202	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 the **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 115
- Importing and exporting tool data (e.g., from a tool presetter or for the programming station)
  Further information: "Importing and exporting of table contents", Page 473
- Table workspace
  Further information: "The Table workspace", Page 462
- Form workspace
  Further information: "The Form workspace for tables", Page 470

Description of function

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

In the tool management, the control displays the parameters of the following tool tables:

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 478
- Turning-tool table **toolturn.trn** (#50 / #4-03-1)
  Further information: "Turning tool table toolturn.trn (#50 / #4-03-1)", Page 488
- Grinding-tool table **toolgrind.grd** (#156 / #4-04-1)
  Further information: "Grinding tool table toolgrind.grd (#156 / #4-04-1)", Page 492
- Dressing-tool table **tooldress.drs** (#156 / #4-04-1)
  Further information: "Dressing tool table tooldress.drs (#156 / #4-04-1)", Page 503
- Touch-probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 506

In the tool management, the control also displays parameters from the **tool_p.tch** pocket table (e.g., **P**).

Further information: "Pocket table tool_p.tch", Page 511

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 191

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.
- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool 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.
- In the **Form** workspace, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.

  Further information: "Adding a column in the workspace", Page 472
- 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 it finds a tool with the entered characters, the control selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window
- 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
- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.
- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.
- In the **Table** workspace of the tool manager, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.

  Further information: "Virtual columns", Page 469

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 (#140 / #5-03-2))

Related topics

- The **Simulation** workspace
  Further information: Programming and Testing User's Manual
- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
  Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
- Adding a tool model to the tool definition (#140 / #5-03-2)
  Further information: "Tool model (#140 / #5-03-2)", Page 209
- 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 611

Requirements

- Kinematics description
  The machine manufacturer creates the kinematics description.
- The insertion point is defined within the machine's kinematic description.
  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 206

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 370

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 207

9.5.1 Assigning a tool carrier

To assign a tool carrier to a tool:

- Select the **Tables** operating mode

- Select the **Tool management** application
- Select the tool you want to use

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

- 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 (#140 / #5-03-2)). The tool carriers are thereby protected from colliding with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
- 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 208

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 (#140 / #5-03-2))

> 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 (#140 / #5-03-2))
  Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
- Tool carrier management
  Further information: "Tool carrier management", Page 205
- 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 611

Requirements

- Collision Monitoring v2 (#140 / #5-03-2) software option
- The tool has been defined in tool management
  Further information: "Tool management ", Page 203
- 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 210
- The tool model has been assigned to the tool
  Further information: "Assigning a tool model", Page 211

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 200

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 370

> For tool models, the same requirements with respect to STL and M3D files apply as for fixtures.
>
> **Further information:** "Options for fixture files", Page 256

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 230
- 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 186

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 the **Tool management** application
- 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 478
- 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 514
- 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 514
- The **Generate tool-usage file** setting is set to **Once** or **Always**
  Further information: "The Machine and Simulation areas", Page 581
- Use the same tool table for the simulation as for 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 **Tests** column
  Further information: Programming and Testing User's Manual

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

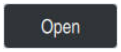
Further information: "Tool usage file", Page 514

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 **Tests** 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 514

- 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 514
- 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 516
- An overview of all tool calls of the NC program active while executing the program is displayed by the control in the **Tooling list** table (#93 / #2-03-1).

  Further information: "Tooling list (#93 / #2-03-1)", Page 518
- 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 218
B-CS	Basic coordinate system basic coordinate system	Page 221
W-CS	Workpiece coordinate system workpiece coordinate system	Page 223
WPL-CS	Working plane coordinate system working plane coordinate system	Page 226
I-CS	Input coordinate system input coordinate system	Page 229
T-CS	Tool coordinate system tool coordinate system	Page 230

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

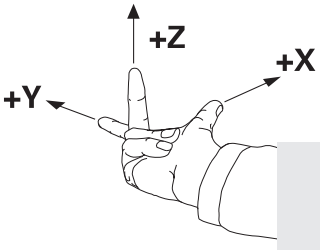
Types of coordinate systems

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.

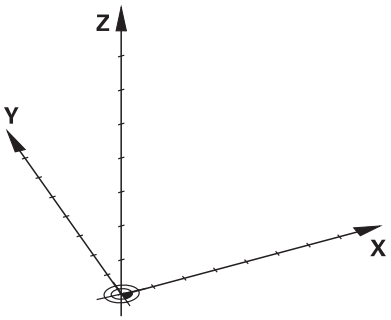
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

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

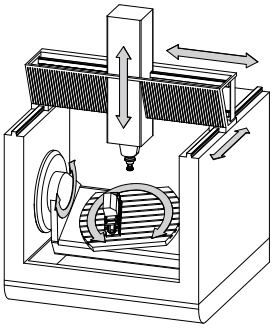
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

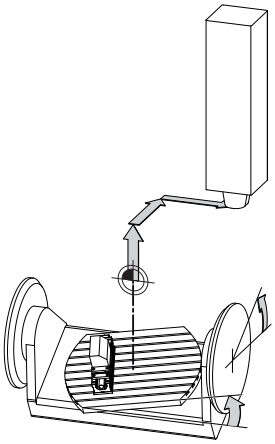
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate 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 520

  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 291

The machine manufacturer can also define further transformations.
Further information: "Note", Page 220

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)**
- **Nominal distance to go (REFDST)**

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**.

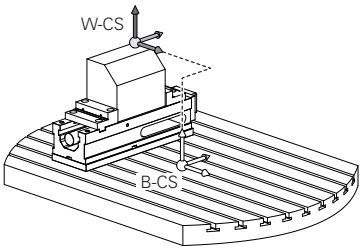
Transformations in the basic coordinate system B-CS

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 233



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

Further information: "Note", Page 222

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.

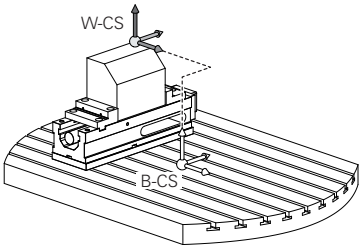
Description of function

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 233



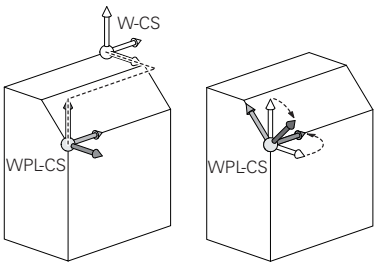
Transformations in the workpiece coordinate system (W-CS)

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 240

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 291

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 229
- 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 226
- 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 229

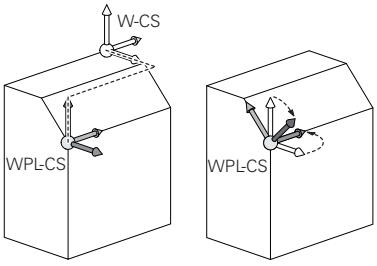
Description of function

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 223

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

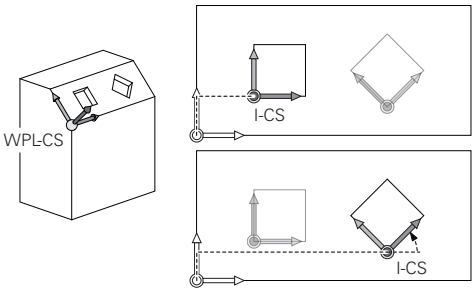


Transformations in the working plane coordinate system WPL-CS

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 (#44 / #1-06-1))

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 291

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 145

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 229
- 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.

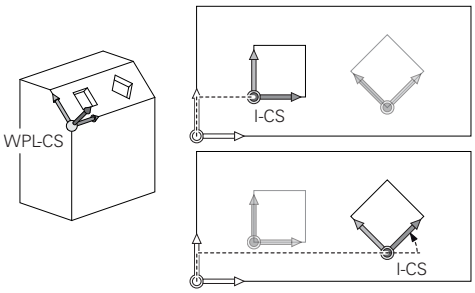
Description of function

Properties of the input coordinate system I-CS

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

Further information: "Working plane coordinate system WPL-CS", Page 226

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 230

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)**
- **Actual distance to go (ACTDST)**

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 226

10.1.8 Tool coordinate system T-CS

Application

The control implements tool compensations and a tool inclination in the tool coordinate system **T-CS**.

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 182

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 185

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 229

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

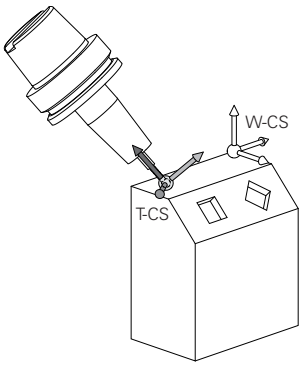
Further information: Programming and Testing User's Manual

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 291

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

Further information: "Contents of display", Page 542

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 182

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 520

Description of function

Setting presets

Presets can be set in the following ways:

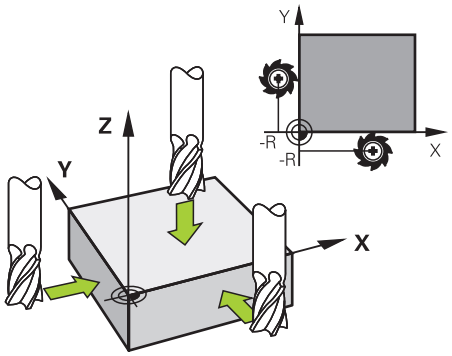
- Setting axis positions manually
  Further information: "Setting a preset manually", Page 236
- Touch probe cycles in the **Setup** application
  Further information: "Touch probe functions in the Manual operating mode", Page 387
- Touch probe cycles in the NC program
  Further information: Measuring 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 525

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, set the preset manually in the desired axis.

Further information: "Setting a preset manually", Page 236

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 237
- 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 223

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.

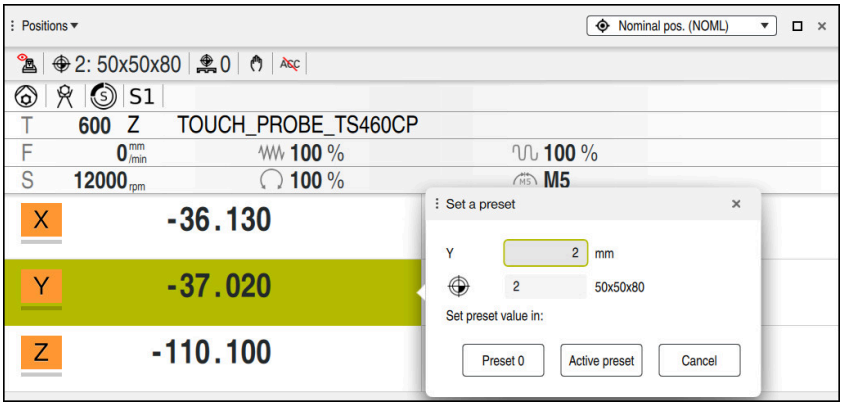
You can use the **3D ROT** (#8 / #1-01-1) button to define whether the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application.

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
- Select **Set the preset**

Set the preset

- 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**)

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.

- 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 127

Further information: "Status overview of the TNC bar", Page 133

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 383
- 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)

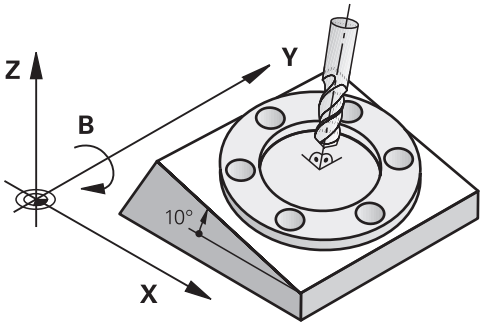
10.3.1 Fundamentals

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 226



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 242
- 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 221

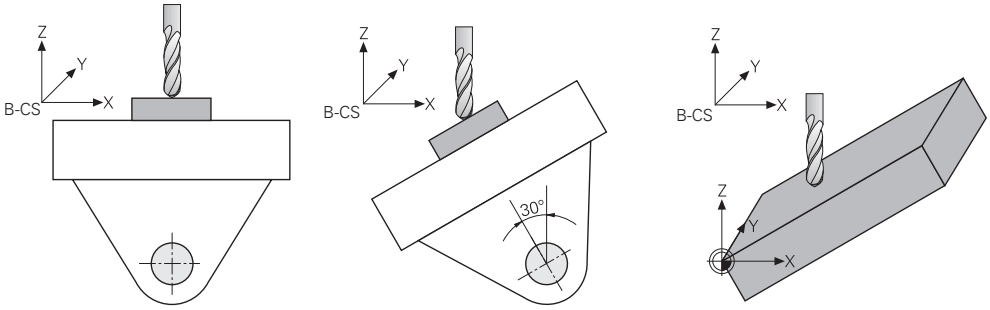
When tilting the working plane, the control moves the machine axes according to the kinematics.

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**.

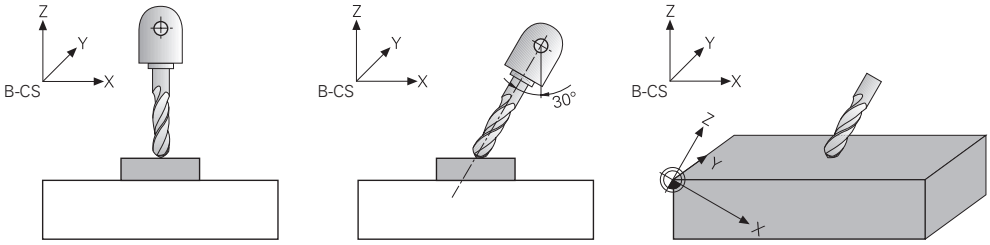
  Further information: "Working plane coordinate system WPL-CS", Page 226



- 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 226



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 422

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

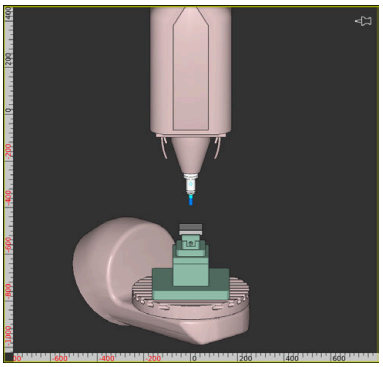
> Even if **M128** or **FUNCTION TCPM** is active with **AXIS POS**, the control will use spatial angles (#9 / #4-01-1) anyway when calculating straight lines **LN** with vectors.

Example: Spatial angles vs. axis angles

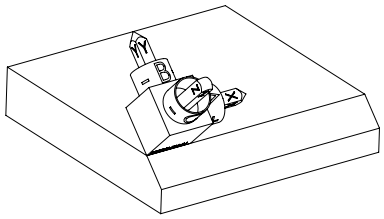
The following example illustrates the difference between spatial angles and axis angles for tilted machining.

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**.



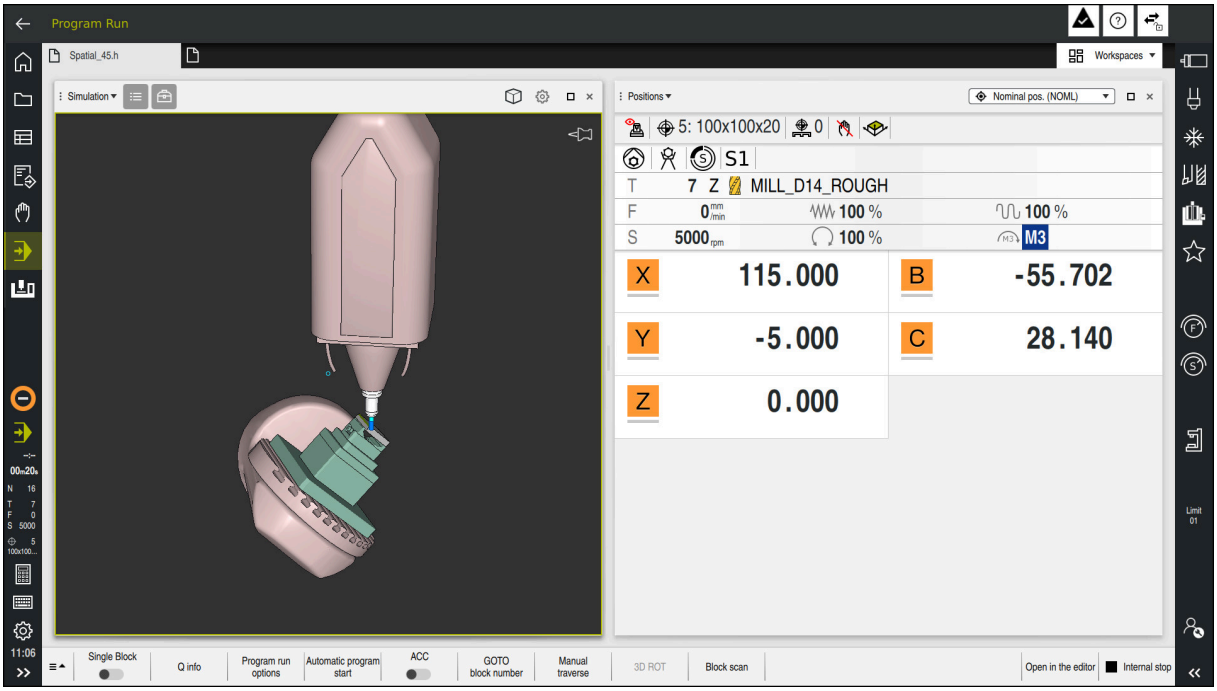
Machine kinematics (not tilted)



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 216

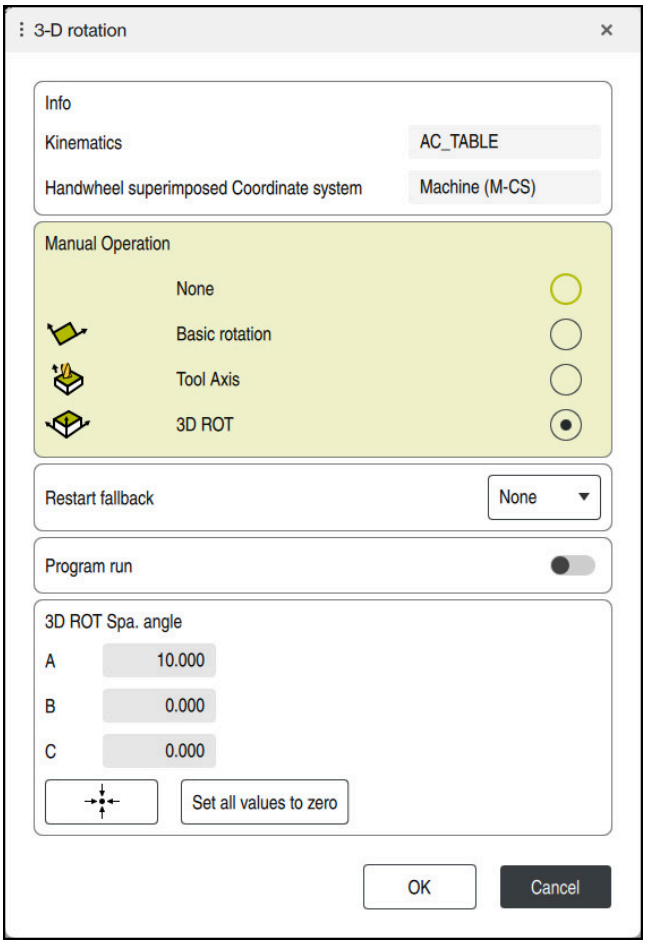
Requirements

- 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 170



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 216 **Further information:** "The Handwheel superimp. function", Page 299 **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 223 ■ **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 245 ■ **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 245 ■ **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 245
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 127

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 235

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

Further information: "Workpiece coordinate system W-CS", Page 223

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 127

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 230

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 226

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

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

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

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **COORD ROT**
  - after **PLANE RESET**
  - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine manufacturer by the machine manufacturer 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 **CfgRotWorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

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

  Further information: "The Referencing workspace", Page 165
- 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

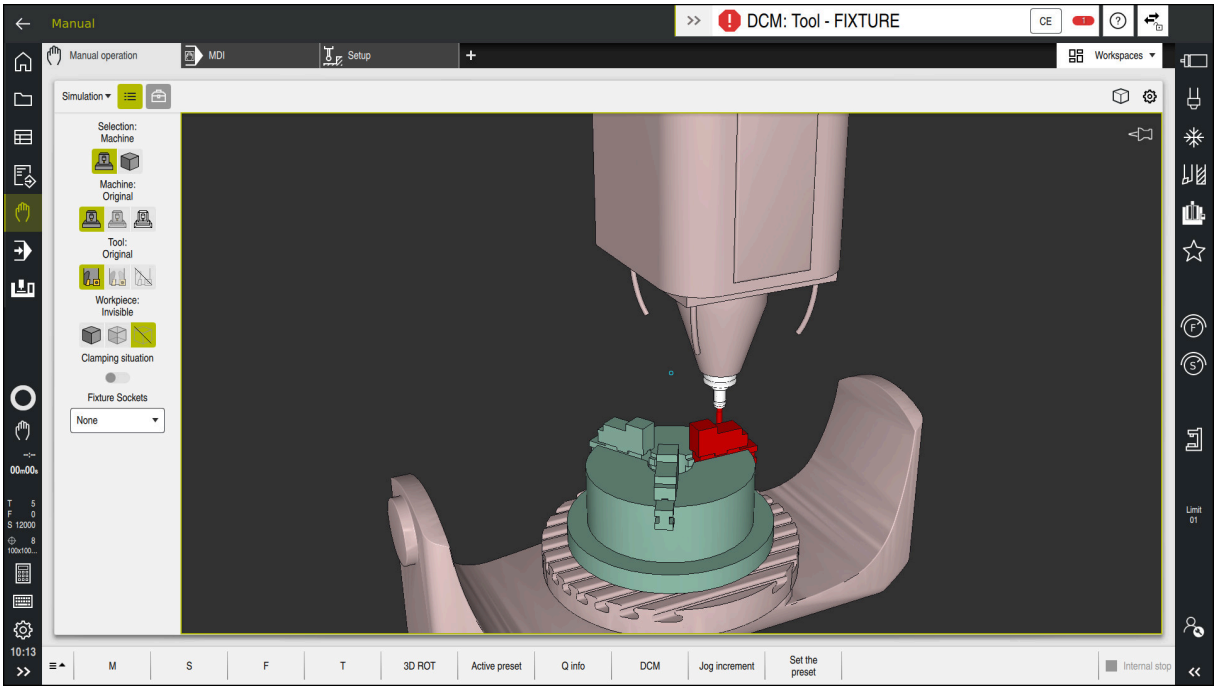
Collision monitoring

11.1 Dynamic Collision Monitoring DCM (#140 / #5-03-2)

Fundamentals

Application

Dynamic Collision Monitoring (DCM, for dynamic collision monitoring) can be used for collision monitoring of fixtures and 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 255
- Integrating fixtures in collision monitoring with the NC function **FIXTURE SELECT**
  Further information: Programming and Testing User's Manual
- Extended tests in the simulation
  Further information: Programming and Testing User's Manual
- Fundamentals of tool carrier management
  Further information: "Tool carrier management", Page 205
- Reduce the minimum clearance between two collision objects (#140 / #5-03-2)
  Further information: Programming and Testing User's Manual

Requirements

- The Collision Monitoring v2 (#140 / #5-03-2) software option
- 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 255
- Fixture already loaded

  For collision monitoring of fixtures, you must load the fixtures with the NC function **FIXTURE SELECT**.

  Further information: Programming and Testing User's Manual
- Tools with a positive radius **R** and length **L**.

  Further information: "Tool table tool.t", Page 478
- The values in the tool management equal the actual tool dimensions

  Further information: "Tool management ", Page 203

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 252

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

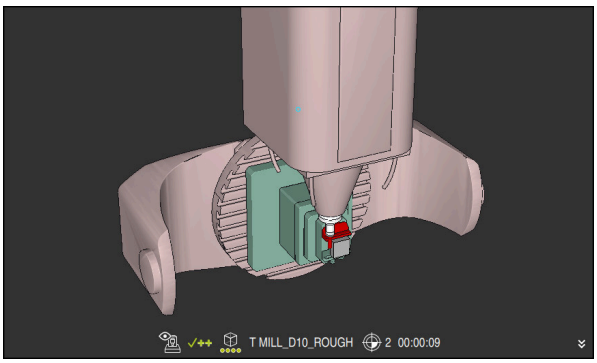
- Select the **Manual** operating mode
- Select the **Manual** application
- Select **DCM**

- > 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 in the control simulation as 3D models for representing fixture situations. You can also represent fixture situations consisting of several individual fixture files.

When DCM is active, the control checks the fixtures for collisions during simulation or machining (#140 / #5-03-2).

Related topics

- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))

  Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 248
- Load fixtures in the NC program with the NC function **FIXTURE SELECT**

  Further information: Programming and Testing User's Manual
- Combine fixtures in the **New Fixture** window.

  Further information: "Combining fixtures in the New Fixture window", Page 276
- Combine fixtures in **KinematicsDesign**

  Further information: "Editing CFG files with KinematicsDesign", Page 268
- 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.
- The insertion point is defined within the machine's kinematic description.

  Refer to your machine manual.

  The machine manufacturer defines the insertion point within the machine's kinematic description. The insertion point defines the position at which the loaded fixtures are placed. The machine manufacturer can define multiple insertion points within a machine's kinematic description (e.g., on a rotary table and on an auxiliary fixture plate).
- Fixtures of suitable format:
  - STL file
    - Max. 20 000 triangles
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file
- If applicable, an insertion point is defined at the fixture

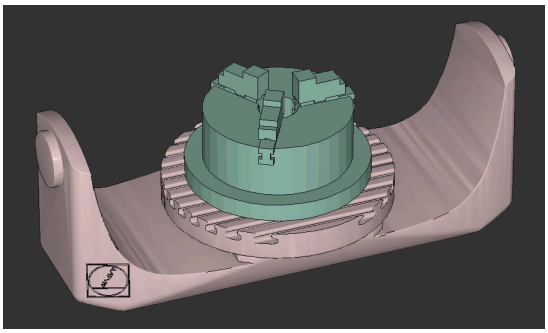
  To combine fixtures with each other in the NC program (#140 / #5-03-2), you must define insertion points in the fixture files. You can define the insertion points in the fixture file in **KinematicsDesign**.

  Further information: "Defining an insertion point", Page 274

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 256
- Fixture placement
  - The **Set up fixtures** function in the **Setup** (#140 / #5-03-2) application
    Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258
  - 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 integrate the fixtures using the **Set up fixtures** function, you can use STL and CFG files (#140 / #5-03-2).

Alternatively, you can manually set up M3D files.

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 370

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 370

Fixtures from CFG files

CFG files are configuration files. You can integrate available STL and M3D files 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 268

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 within the machine kinematics.
- Create a central directory for your fixtures (e.g., **TNC:\system\Fixture**).
- When DCM is active, the control checks the fixtures for collisions during simulation or machining (#140 / #5-03-2).

  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 Measuring the position of a fixture with Set up fixtures (#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 (#140 / #5-03-2)", Page 248
- Fixture monitoring
  Further information: "Fixture management", Page 255
- 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 411

Requirements

- Collision Monitoring v2 (#140 / #5-03-2) software option
- Kinematics description
  The machine manufacturer creates the kinematics description.
- The insertion point is defined within the machine's kinematic description.
  Refer to your machine manual.
  The machine manufacturer defines the insertion point within the machine kinematics. The insertion point defines the position where loaded fixtures are placed.
- Workpiece touch probe
- Permitted fixture file matching the real fixture
  Further information: "Options for fixture files", Page 256

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 263

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:

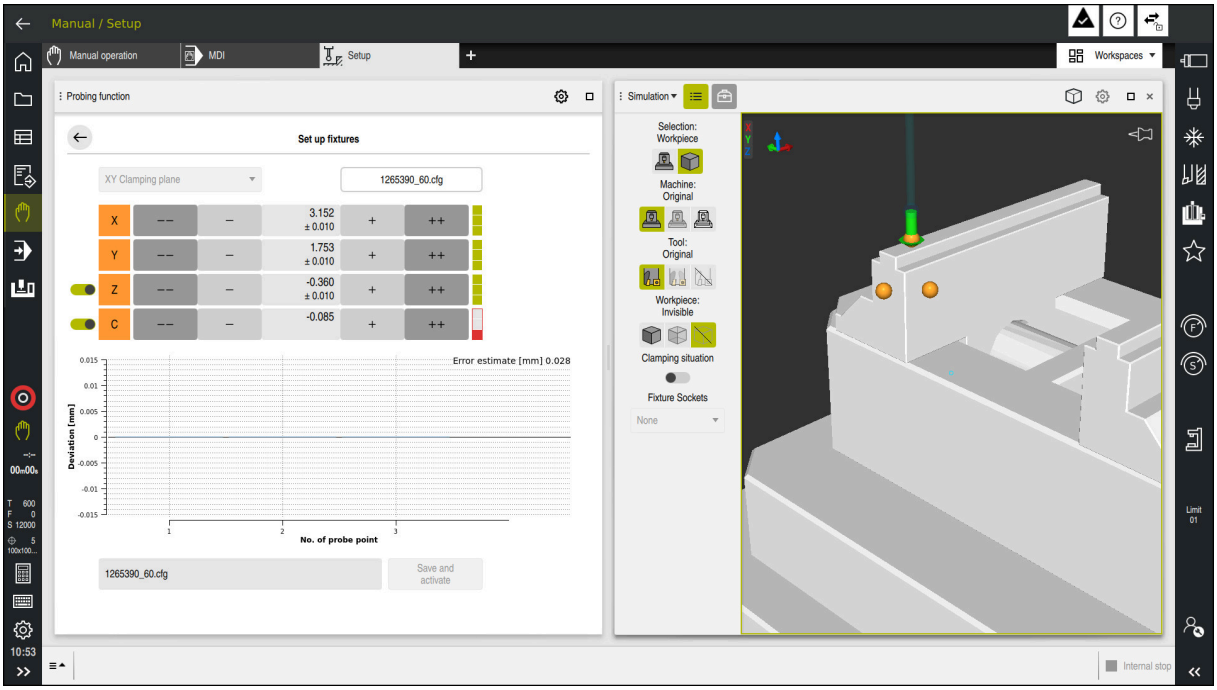
  You cannot tilt before probing. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 238

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.

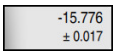
    > Probing on edges, corners or heavily curved fixture areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    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**.**
	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
	■ 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.

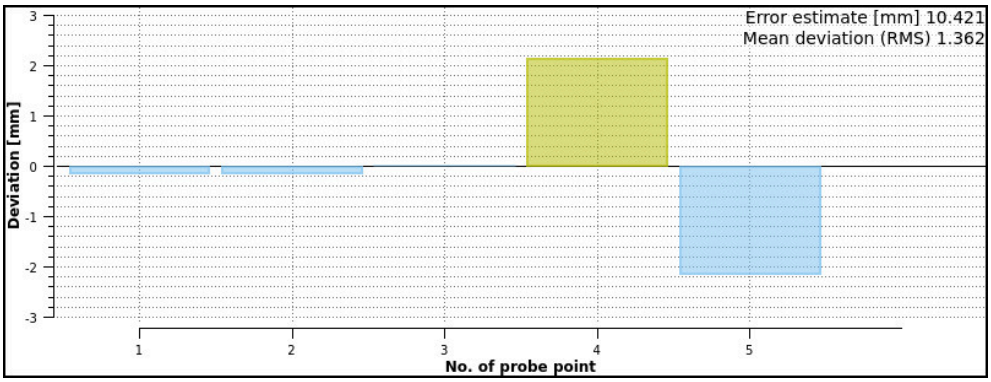
Icon or button	Meaning
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM). When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by **Save and activate** at the end of the measuring process. If you enter a different file name next to the button, the CFG file will not be overwritten. If the fixture has already been loaded at another insertion point within the machine kinematics, the control will display the **Fixture already loaded** window. In this case you must enter a new file name.

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.

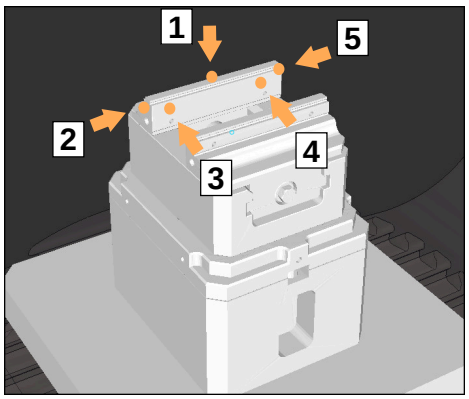
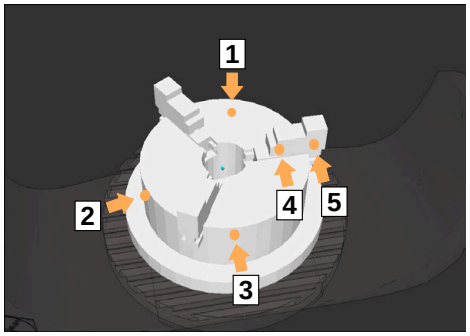
  Once the columns in the error estimate diagram are no longer transparent and the **Error estimate [mm]** value displays the required precision, the set-up process is completed.

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

Machine kinematics with several insertion points (#140 / #5-03-2)

A machine may offer several placing options for fixtures (e.g., rotary tables and auxiliary fixture plate). The machine manufacturer may define insertion points within the machine kinematics model for each placing option. For each machine kinematics model, the control offers only one insertion point where you can measure fixtures. Your machine manufacturer must make a dedicated kinematics model for each insertion point where a fixture is to be measured. Prior to measuring, you must select the right machine kinematics model (e.g., with the **FUNCTION MODE** function).

Further information: Programming and Testing User's Manual

Measuring the fixed-jaw vise

> The desired 3D model must meet the requirements of the control.
> **Further information:** "Options for fixture files", Page 256

To measure a vise using the **Set up fixtures** function:

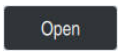
- Affix a real vise in the working space

- 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 probed position. The control marks the probed position with a dot in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
- > The status of the axes turns green.
- Probe another point in axis direction **Y+** for the basic rotation

> To achieve maximum accuracy when probing 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.
- Probe 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 268
- 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.
- The control can measure fixtures only at one insertion point. However, you can measure any number of fixtures and save them for later use.
- When measuring a jaw chuck, the coordinates of the **Z**, **X**, and **Y** axes must be determined just as when measuring a vise. The rotation is determined from one or several jaws.
- 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
- Prior to measuring, check if the correct machine kinematics model is active.

Notes on combined fixtures

- When combining fixtures with the **KinematicsDesign** function or the **New Fixture** window, you can measure these fixtures.

  Further information: "Integrating 3D models", Page 273

  Further information: "Combining fixtures in the New Fixture window", Page 276
- When combining fixtures with the **FIXTURE SELECT** function in the NC program, you can measure only the fixture at the insertion point of the machine.

  You can measure a datum fixture system, for example. All fixtures loaded on this system will be oriented to this datum fixture system.

  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.

If a fixture is not available as a CFG file, the control offers the following options:

- Integrate the 3D model of the fixture in **KinematicsDesign** (e.g., STL file)
  Further information: "Integrating 3D models", Page 273
- Save the fixture as a CFG file with the **Combine fixtures** function
  Further information: "Combining fixtures in the New Fixture window", Page 276

Related topics

- Combine fixtures into complex clamping arrangements
  Further information: "Combining fixtures in the New Fixture window", Page 276

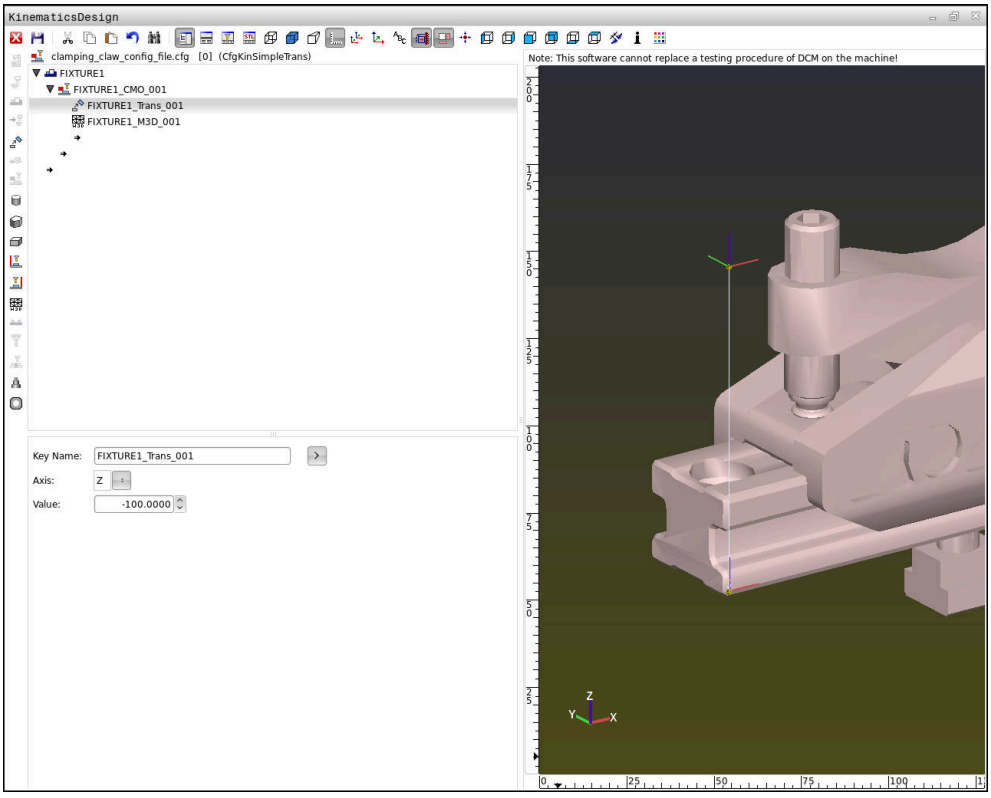
Description of function

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
- Definition of insertion points for fixtures

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:

Syntax element	Meaning
`key:= ""`	Name of the function
`dir:= ""`	Direction of a transformation (e.g., **X**)
`val:= ""`	Value
`name:= ""`	■ Name displayed by the control in case of collision Optional syntax element ■ Name of the insertion point
`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	Meaning
`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

Function	Meaning
`CfgCMO ( key:="fixture", primitives:= ["XShiftFixture","CRot0", "Fixture_body"], active :=TRUE, name :="")`	Describes all of the transformations contained in the fixture. The parameter active **:= TRUE** activates collision monitoring for the fixture. The **CfgCMO** contains collision objects and transformations. The fixture is combined based on the arrangement of the different transformations. Here, the transformation **XShiftFixture** shifts the center of rotation of the transformation **CRot0**.
`CfgKinFixModel(key:="Fix_Model", kinObjects:=["fixture"])`	Fixture designation **CfgKinFixModel** contains one or more **CfgCMO** elements.
`CfgKinFixSocket(key:="Fix_Socket", name:="Fix_Socket")`	Insertion point for fixtures The **CfgKinFixSocket** contains an insertion point. Each insertion point must be given an unambiguous name.

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 key name for the fixture (e.g., FIX_clamping jaw)
- Confirm your input
- > **KinematicsDesign** applies the input to the fixture entry.

- Open the fixture entry

- Move the cursor down one level

- Select **Insert collision object**
- > **KinematicsDesign** creates a new collision object.
- Enter a key name for the collision object (e.g., COL_clamping jaw)
- Enter a name for the collision object (e.g., clamping jaw)
- Confirm your input
- > **KinematicsDesign** applies the input and a fixture entry with a collision object has now been created.

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 **Insertion point** under Collision object

- Select the desired geometric shape (e.g., a cuboid)
- > **KinematicsDesign** inserts the shape.
- 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
- > **KinematicsDesign** applies the input and displays the defined cuboid in the graphics.

Integrating 3D models

The integrated 3D models must meet the requirements of the control.

Further information: "Options for fixture files", Page 256

To integrate a 3D model as a fixture:

- Create a fixture entry with a collision object

- Select **Insertion point** under Collision object

- Select **Insert 3D model**
- > **KinematicsDesign** opens the **Open File** window.
- Select the desired STL or M3D file
- Select **OK**
- > **KinematicsDesign** 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**:

- Creating a fixture entry with a collision object

- Select **Insertion point** under Collision object

- Select **Insert transformation**
- > **KinematicsDesign** inserts the transformation and displays it in the graphics.
- Enter a key name for the transformation (e.g., Z shift)
- Select the axis for the transformation (e.g., **Z**)
- Enter the value for the transformation (e.g., **100**)
- Confirm your input
- > **KinematicsDesign** takes on entering this and the fixture has now been placed.

Defining an insertion point

The content below describes the procedure with a fixture of the ***.cfg** file type.

To define an insertion point:

- Open the fixture with **KinematicsDesign**

- Open the fixture entry

- Move the cursor down one level

- Open the collision object

- Select **Insertion point**

- Select **Insert transformation** if required (e.g., **Z+**)

> The insertion point for combining fixtures is defined in the NC program. Shift the insertion point to the desired position by means of transformations.

> **KinematicsDesign** inserts the transformation and displays it in the graphics.

- Enter a key name for the transformation (e.g., Z shift)
- Select the axis for the transformation (e.g., **Z**)
- Enter the value for the transformation (e.g., **100**)
- Confirm your input

> **KinematicsDesign** loads the input.

- Select **Insert insertion point**

> **KinematicsDesign** inserts the insertion point and displays it in the graphics.

- Enter the key name for the insertion point (e.g., FIX_1_GP)
- Enter a name for the insertion point (e.g., insertion point _1_base plate)

> Every insertion point must have an unambiguous name.

> **KinematicsDesign** applies the input and the insertion point has now been defined.

- Insert further transformations if required (e.g., **X+**)
- Define another insertion point as needed

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 276
- 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.
- You can create a CFG file in the **Quick selection new file** workspace.
  Further information: Programming and Testing User's Manual

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 255
- Integrating fixtures into the NC program
  Further information: Programming and Testing User's Manual
- Set up fixtures (#140 / #5-03-2)
  Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258

Requirement

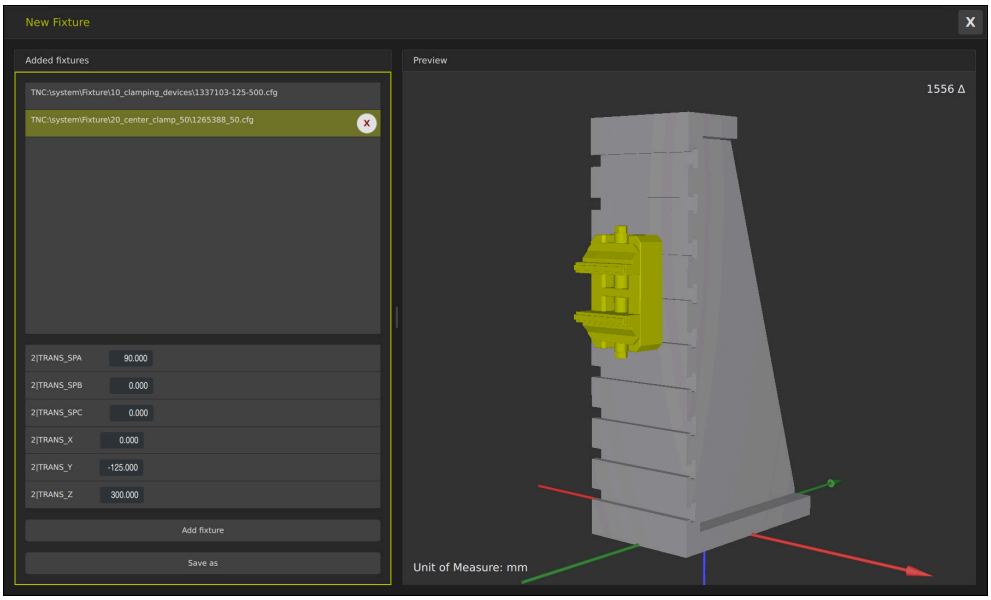
- Fixtures of suitable format:
  - STL file
    - Max. 20 000 triangles
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

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 268
- The **Combine fixtures** function allows saving individual fixtures of ***.stl** or ***.m3d** file types as CFG files. You need a fixture as a CFG files when you wish to define insertion points, for example.

  Further information: "Defining an insertion point", Page 274

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 528

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 528
- Define settings for AFC for each tool in the tool management
  Further information: "Tool table tool.t", Page 478
- Define AFC in the NC program
  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 283
- 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 284
- Prior to automatic control, determine the reference spindle power with a teach-in cut
  Further information: "AFC teach-in cut", Page 286

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 127

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 137

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 288
- 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 528
- ***.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 530
- ***.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 performing 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 532

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 533

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 528
  - 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 288
- 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 528
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 478
- **AFC** toggle switch active

  Further information: "The AFC toggle switch in the Program Run operating mode", Page 284

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 283

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 127

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 damage 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.
- The feed rate override is the percentage value of the programmed feed rate that you usually set with the potentiometer. When the adaptive feed control is active in the **Control** mode, the control defines the feed rate override.
  - Turning the feed rate potentiometer will not influence the feed rate override.
  - 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 478
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function
  Further information: "NC functions for AFC (#45 / #2-31-1)", Page 283

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 528
- Desired control setting defined for all tools
  Further information: "Tool table tool.t", Page 478
- 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 284

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 530

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 286
- Tables for AFC
  Further information: "Tables for AFC (#45 / #2-31-1)", Page 528

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 528
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 530
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 532
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 286 ■ 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 530 ■ 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 127

> 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 478

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 528

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 528

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 286

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 478

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 426

Further information: "The MDI application ", Page 383

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

Further information: "The Positions workspace", Page 127

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 141
- Reference systems of the control
  Further information: "Reference systems", Page 216

Requirement

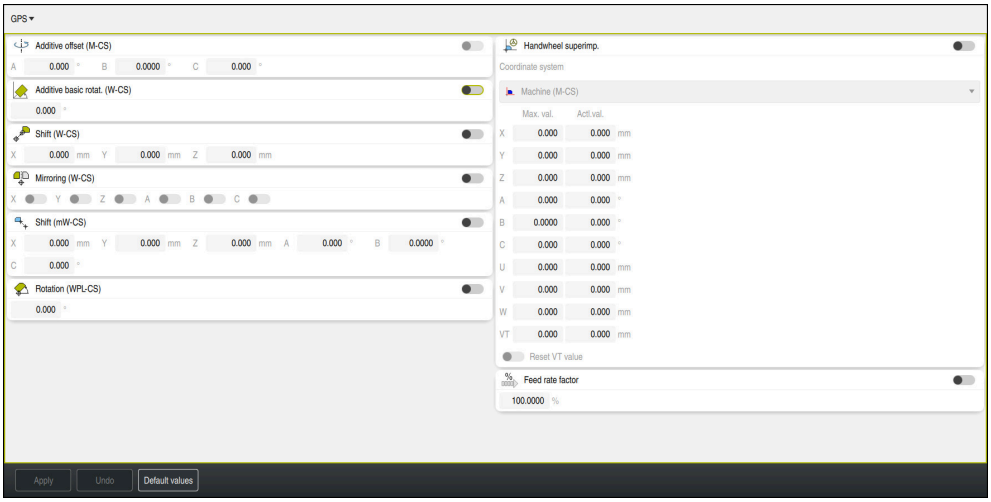
- Software option Global PGM Settings (#44 / #1-06-1)

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 141

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)

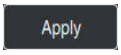
The Global Program Settings (GPS) include the following functions:

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 294
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 296
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 296
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system **W-CS** **Further information:** "The Mirroring (W-CS) function", Page 297
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 298
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 299
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel **Further information:** "The Handwheel superimp. function", Page 299
Feed rate factor	Manipulation of the active feed rate **Further information:** "The Feed rate factor function ", Page 302

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°**)



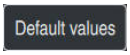
- Press **Apply**
- > The control accepts the entered values.

> 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):

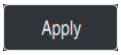
- Select an operating mode (e.g., **Program Run**)
- Open the **GPS** workspace



- 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 218
- Difference between basic rotation and offset
  Further information: "Basic transformation and offset", Page 522

Description of function

The control adds the value to the active axis-specific offset from the preset table.

Further information: "Preset table *.pr", Page 520

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 127

Application example

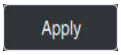
The travel range of a machine with AC fork head is increased using the **Additive offset (M-CS)** function. An eccentric tool chuck is used and the zero position of the C axis is shifted by 180°.

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 **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

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 520

The **Additive basic rotat. (W-CS)** function has no effect on the position display.

Application example

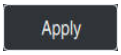
You rotate the CAM output of an NC program by 90° and compensate for the rotation using the **Additive basic rotat. (W-CS)** function.

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 223

The **Shift (W-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 158

Application example

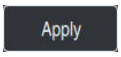
The surface of a workpiece to be reworked is determined using the handwheel and the offset is compensated for using the **Shift (W-CS)** function.

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.** toggle 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**
- Starting 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 158

Application example

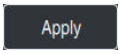
The **Mirroring (W-CS)** function makes the control carry out machining in a mirror-inverted way.

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 223

The **Shift (mW-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 158

A modified workpiece coordinate system **mW-CS** is present with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without these preceding coordinate transformations, the **Shift (mW-CS)** option would be effective directly in the workpiece coordinate system (**W-CS**) and would thus be identical to **Shift (W-CS)**.

Application example

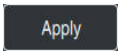
You mirror the CAM output of an NC program. After mirroring, you shift the workpiece datum in the mirrored coordinate system to produce the counterpart to a mirror cap.

Initial situation:

- Available CAM output for right mirror cap
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **Function TCPM** function with spatial angles output
- The left mirror cap is to be machined

To shift the datum in the mirrored coordinate system:

- Open the **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 446

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 127

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 147

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 239

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 147

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 (#140 / #5-03-2), 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 (#140 / #5-03-2)", Page 248
- **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 127

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 431

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 129

13

Process monitoring (#168 / #5-01-1)

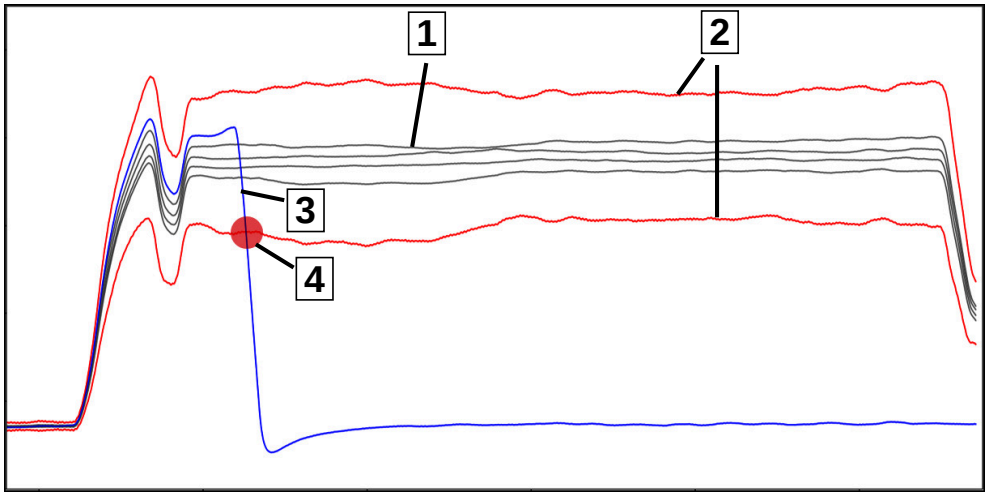
13.1 Fundamentals

The control uses process monitoring to detect process problems and provides information for you to improve productivity.

Process monitoring assists you in the following applications:

- Identifying tool or cutting edge breakages
- Identifying tool wear
- Detecting dimensional deviations during machining
- Identifying set-up problems (e.g., faulty or missing pre-machining or wrong materials)
- Identifying machining with feed-rate override and spindle override that is either too high or too low

Process monitoring compares the signal curves 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 torque drops due to tool breakage.



Example: Torque drop due to tool breakage

1 — Recorded machining operations

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 operation. If the spindle load is too low, the control may not detect a difference from idling (for example, 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 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 is to monitor (e.g., the torque). The machine provides information about the machining process by means of signals.
Procedure	The method defines how the control will monitor the signal (e.g., constant limits).
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.
Recordings	The control records machining operations and displays them in the **Recordings - Table** area. If you assess recordings as being "good parts," the control will use the first ten of 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 course for the monitoring tasks.

Process monitoring in earlier software versions is not compatible with software version 20. Make sure to delete old recordings when updating the software. The monitoring tasks must be newly set up and new recordings must be made.

13.2 First steps in process monitoring

13.2.1 Starting process monitoring

> Activate process monitoring only after proving-out the workpiece, when the monitored sections of the NC program no longer change. Do not change the feed-rate potentiometer setting during a monitoring process.

Start process monitoring as follows:

- ▶ Open the NC program in the **Editor** operating mode
- ▶ Define the start of a monitoring section with **MONITORING SECTION START**
- ▶ Define the end of a monitoring section with **MONITORING SECTION STOP**
- ▶ Select **Select in Program Run**

  Select in Program Run
- > The control opens the NC program in the **Program Run** operating mode.
- ▶ Open the **Process Monitoring** workspace
- > The control displays the process monitoring start page.
- > The control marks a pre-selection of monitoring tasks.
- ▶ Select **Activate**

  Activate
- > The control activates process monitoring with the selected monitoring tasks 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** method 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 **Recordings - Table** area.
- > 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** method. Depending on the monitoring task, several assessments may be required before monitoring is active.
>
> **Further information:** "Procedure", Page 311

- ▶ Machine further workpieces
- ▶ If applicable, assess the parts resulting from the machining operations in the **Evaluate component** area

You can adapt the parameters of the monitoring tasks to the machining operation as needed.
Further information: "Modifying the parametrization of monitoring tasks", Page 307

13.2.2 Modifying the parametrization of monitoring tasks

To modify the parametrization of monitoring tasks:

- Enable the **Runtime** mode as needed
- Select the desired monitoring section from the right part of the **Recordings - Table** area
- If applicable, open the **Form** column that contains the monitoring tasks
  > In the **Form** column, the control displays the monitoring tasks including the recorded machining operations as graphs.
- Open **Settings** within the monitoring task for parameterizing
  > The control displays the signal and the settings of the selected recording.
- Adapt the sliders in the **Parameter settings** area
- Select **Expand**, if necessary
  > The control shows the parameters that were active at the time of the marked recording. You can compare the values.
- Adapt the **Reactions**, if required
- Select **OK**
  > The control saves the changes and activates them when the NC program is executed the next time.

The parameters and reactions can also be modified in the **Setup** mode. This option allows modifying the parameters of a monitoring task in all monitoring sections at the same time, but you cannot compare the values.
Further information: "The Form column in Setup mode", Page 325

13.2.3 Changing the monitoring task

To change a monitoring task:

- Enable the **Setup** mode as needed
- > 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

Change

- Select **Settings** in the **Monitoring task** area
- > The control opens the **Monitoring task** window with the same table as on the start page.
- > The control displays all available monitoring tasks as rows and the applications as columns.
- Select a new monitoring task for the desired application

OK

- Select **OK**
- > The control saves your change.

> You can modify each monitoring task separately in the **Runtime** mode.
> **Further information:** "Graphic display of the monitoring tasks", Page 332

13.2.4 Removing a monitoring task

To remove a monitoring task:

- Enable the **Setup** mode as needed
- > 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 from all monitoring sections shown.

- If applicable, open the **Form** column

Remove

- Select **Remove** in the **Monitoring task** area
- > The control opens a window with a confirmation prompt.

Yes

- Select **Yes**
- > 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.
> - You can remove each monitoring task separately in the **Runtime** mode.
>
> **Further information:** "Graphic display of the monitoring tasks", Page 332

13.3 Monitoring tasks

A monitoring task consists of the following properties:

- Signal (e.g., **Spindle current (smoothed)**)
- Method 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 which are suitable for different applications.

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. In this process, you assign each monitoring task to an application

13.3.1 Signals

Process monitoring offers the following signals, which are suitable for different applications:

Signal	Meaning	
Unexpected servo lag	Explanation:	Parameter indicating by how much the following error of the linear axes deviates from the expected value
	Tip:	High values indicate tool or cutting edge breakage. You can use this signal for tools with a diameter of 3.8 mm or greater. This signal does not need any reference machining.
Lag perpendicular to path	Explanation:	Deviation of movement vertical to the calculated path in µm
	Tip:	When the monitoring task **Perpendicular servo lag — Tunnel** shows a large value, this indicates a dimensional deviation (e.g., due to tool wear). You can define the tunnel width to suit your tolerances.
Torque Only on machines with GEN3	Explanation:	Calculated spindle torque in Nm Shows when the tool is cutting
	Tip:	Use this signal instead of **Spindle current (smoothed)**. To enable precise comparison of the machining operations, use a method with reference machining.
Spindle current (smoothed)	Explanation:	Spindle current in A Shows when the tool is cutting
	Tip:	To enable precise comparison of the machining operations, use a method with reference machining.
Lag parallel to path	Explanation:	Deviation of movement in parallel to the calculated path in µm
Feed-rate override	Explanation:	Current value of feed rate override in %
	Tip:	This signal does not need any reference machining.
Spindle override	Explanation:	Current value of spindle override in %
	Tip:	This signal does not need any reference machining.
Tooth feed rate Fz	Explanation:	Current feed rate per tooth
	Tip:	You can use this display only if a number of teeth **CUT** is defined for the tool.
Testing signal (no compatibility)	Explanation:	The signal may change between different software statuses.
	Tip:	This monitoring task is intended for test purposes and should be used only if requested by HEIDENHAIN or by the machine manufacturer!

13.3.2 Procedure

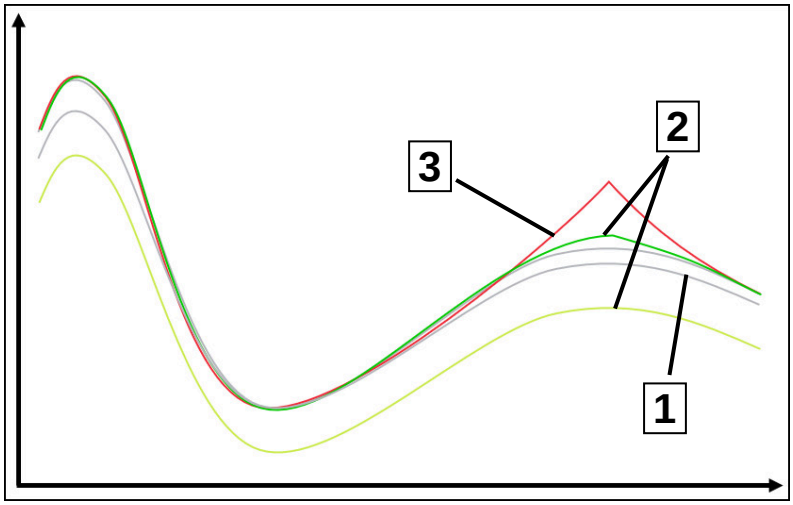
Process monitoring offers the following procedures:

- **Waveform comparison**
  Further information: "Waveform comparison", Page 311
- **Tunnel**
  Further information: "Tunnel", Page 312
- **Display**
  Further information: "Display", Page 313
- **Constant**
  Further information: "Constant", Page 313

Waveform comparison

In the **Waveform comparison** method, the control compares the current signal wave with the recordings of "good parts" at short time intervals. If the waveform deviates too much, the monitoring task identifies a potential fault. A long-term signal drift of a constant signal offset will not modify the waveform and will therefore not trigger 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.

Both machining operations do not trigger a reaction.

3 Machining with a heavy deviation

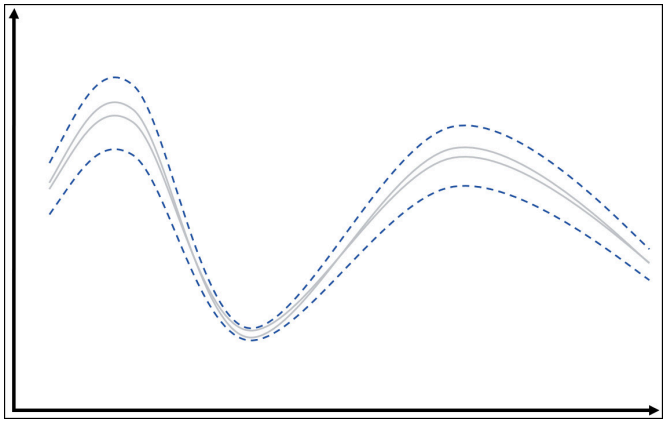
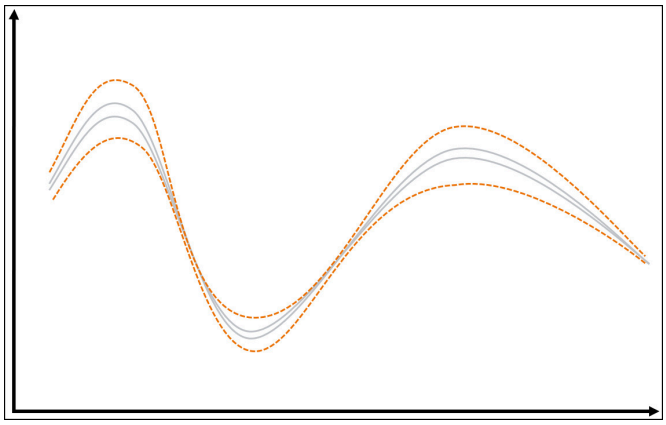
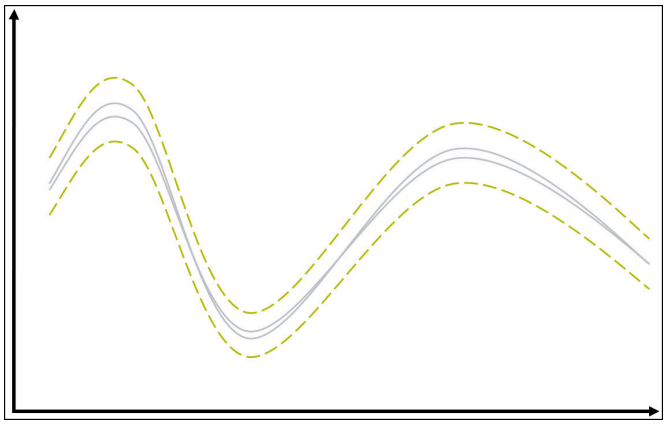
The waveform of this machining operation deviates heavily from the previous records and will trigger the configured reactions.

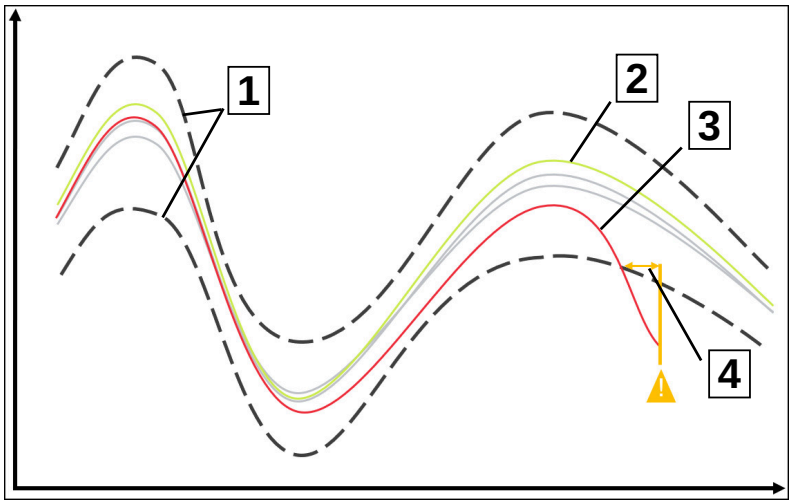
Tunnel

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
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)	
Absolute, static tolerance (e.g., tunnel width 30 A) The gray lines represent the reference machining operations	



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 defined holding time exceeds the upper or lower error limit, the control triggers the reactions.

Display

In the **Display** method, 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** method, the control monitors if the current machining operation 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 monitor with this method starting from the first machining operation, and does not require any evaluation of recordings. If the defined holding time exceeds the upper or lower error limit, the control triggers the reactions.

13.3.3 Parameters

The parameters of the monitoring tasks define how sensitively process monitoring reacts.

Depending on the method, the monitoring tasks contain the following parameters:

Method	Parameter
Waveform comparison	Tolerance of waveform relative to reference machining operations in %
Tunnel	■ Tolerance percentage relative to reference machining operations in % ■ Multiple of measured standard deviation σ of the reference machining operations ■ Static tunnel width in the signal unit ■ Hold time for reactions in ms
Display	No parameters
Constant	■ Upper limit in the signal unit ■ Possibly lower limit in the signal unit ■ Hold time for reactions in ms

You can change the parameters at the following points of the **Process Monitoring** workspace:

- For several monitoring tasks at the same time
  Further information: "The Form column in Setup mode", Page 325
- For each monitoring task individually, including comparison of parameters before and after
  Further information: "Settings for parameterizing of monitoring tasks", Page 335
- During machining operation by means of the **Parameter learning** function
  Further information: "Parameter learning", Page 344

13.3.4 Reactions

If a signal exceeds the error limits for longer than the defined hold time, the monitoring task may trigger one or more reactions.

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 379
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 203

You can define the reactions in the following locations of the **Process Monitoring** workspace:

- The **Form** column in **Setup** mode
  Further information: "The Form column in Setup mode", Page 325
- Settings for the parameterization of each monitoring task individually
  Further information: "Settings for parameterizing of monitoring tasks", Page 335

Refer to your machine manual.
The machine manufacturer can define further reactions.

13.3.5 Applications

The control provides various cases that you detect or monitor with the monitoring tasks.

The signals and methods deliver values which are meaningful for different cases. When you select monitoring tasks, the control shows how well the monitoring tasks are suited for which cases.

You can assign the monitoring tasks to the following cases by default upon selecting:

Application example	Example
Tool or tooth breakage	Detects tool or cutting edge breakages (e.g., to change to a replacement tool)
Tool wear	Detects tool wear relative to the reference machining operation (e.g., in order to replace the tool on the basis of this data)
Accuracy	Detects dimensional deviations during milling operations relative to the reference machining operation (e.g., as the result of tool displacement) Process monitoring cannot consider all influences on accuracy (e.g., temperature variations).
Setup problems	It identifies, for example, incorrect fixture situations, missing or faulty pre-machining, and missing or faulty tools relative to the reference machining operation.
Incorrect override settings	Detects deviations from the defined feed rate and spindle override limits (e.g., potentiometer settings modified by mistake).

To simplify the assignment, the control displays the case and the name of the monitoring task within process monitoring.

13.4 Process monitoring start page

Application

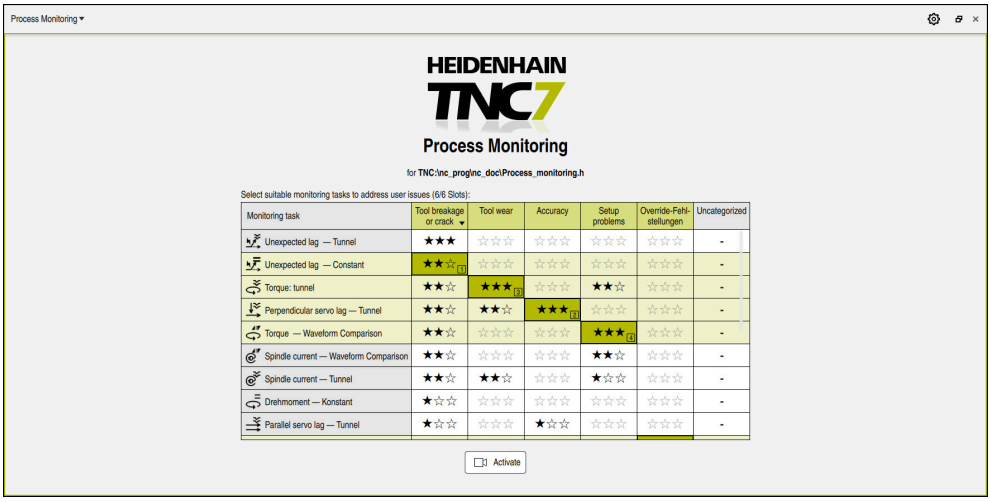
When opening the **Process Monitoring** workspace for the first time for a NC program, the control displays a start page.

A table in the control shows all available monitoring tasks and how well they are suited for which cases. The control suggests a pre-selection of monitoring tasks, and you can modify this selection.

Description of function

When resetting the monitoring settings completely, the control displays the start page again.

The control displays the path of the selected NC program and a table for selecting the monitoring tasks including the cases for the NC program.



Start page of the **Process Monitoring** workspace

Selecting the monitoring tasks

The control displays a table with all available monitoring tasks in rows and possible process-monitoring use cases in columns. The stars in the cells indicate how well suited the monitoring task is for the corresponding use cases.

The table also includes the monitoring tasks defined by the machine manufacturer.

When you select the NC program anew, the control will preselect six monitoring tasks. Each use case is selected at least once. You can change the selected items. If you reset the monitoring settings, the control will mark all monitoring tasks that had been marked when process monitoring was last activated before the reset.

Each row in the table is a combination of a monitoring task and a use case. The control displays the selected use case in process monitoring.

If a monitoring task has not been defined as suitable for a use case, you can select the **Not categorized** use case for it.

Operation

- You can sort the rows for a use case by tapping or clicking the desired cell in the header.
- If you long-press or right-click a header, the control will open a pop-up window that contains further details on the use cases.
- The column order can be changed by dragging a header cell to the desired position.
- If you tap or click a marked cell, the control will unmark it.
- The control displays numbers from 1 to 6 in the cells of marked monitoring tasks. They indicate the order in which the monitoring tasks will be displayed in process monitoring.

13.5 The Process Monitoring workspace (#168 / #5-01-1)

Application

The **Process Monitoring** workspace contains all options for setting up and using process monitoring.

The general area of the **Process Monitoring** workspace is available in all modes. You can activate, deactivate, or reset process monitoring. The control displays information regarding process monitoring.

The other sections of the workspace depend on which mode is active — **Setup** or **Runtime**.

Related topics

- Process monitoring start page
  Further information: "Process monitoring start page", Page 317

Requirements

- Software option Process Monitoring (#168 / #5-01-1)
- Reproducible machining defined as monitoring section
  Further information: Programming and Testing User's Manual
- Program run in Full Sequence mode
- The **FUNCTION MODE MILL** milling mode

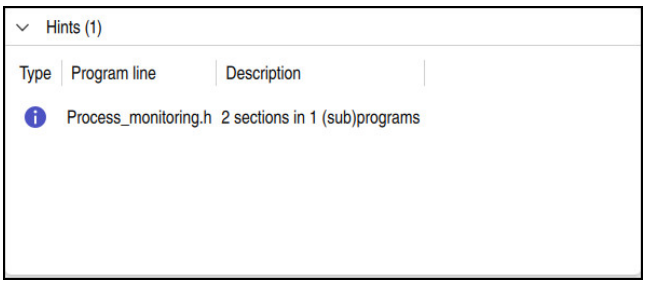
Description of function

The **Process Monitoring** workspace is available in the **Program Run** operating mode.

The title bar of the **Process Monitoring** workspace includes the following icons:

Icon	Meaning
	Process monitoring mode:
	The **Setup** mode **Further information:** "The Setup mode", Page 322
	The **Runtime** mode **Further information:** "The Runtime mode", Page 326
	Open or close the **Filter** column The content of this column depends on the active mode.
	Open or close the **Form** column The content of this column depends on the active mode.
	Open or close global **Settings** **Further information:** "The Settings window", Page 336

General area



General area

The general area contains the following:

- Toggle switch for activating process monitoring for this NC program
- **Reset** icon for resetting certain monitoring settings for this NC program
- **Hints** area regarding the active NC program

Resetting

When selecting the **Reset** icon, the control opens the **Reset** window.

The control resets the following process monitoring settings to the default value:

- Selected monitoring tasks

  Resets the monitoring tasks including their parameters and reactions to the value defined on the start page.
- Columns in the **Recordings - Table** area
- **Recording strategy** in the **Form** column when **Runtime** mode is active

The **Reset only some of the settings** toggle switch allows you to select which of the three settings the control will reset.

When the toggle is inactive, after resetting the control displays the start page again with the last specified monitoring tasks.

> To view the default pre-selection on the start page, reset process monitoring and re-select the NC program in the **Program Run** operating mode.

The Hints area

The **Hints** area contains the following information:

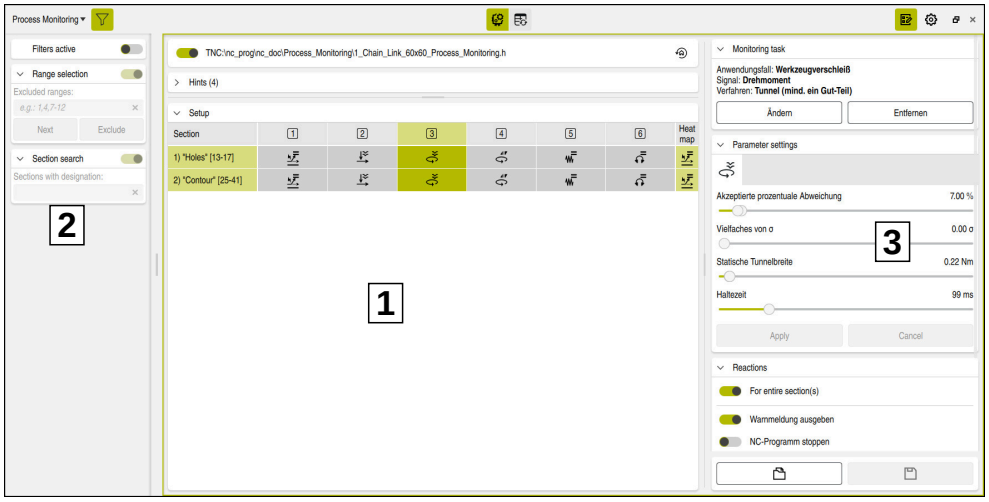
Column or icon	Meaning
Type	In the **Type** column, the control displays various types of notes.
	Information (e.g., the number of monitoring sections)
	Suggestions (e.g., **Consider deleting all records for NC program**) If you change the positioning blocks within a monitoring section, the control can no longer consider the recordings made so far. You must delete the recordings in the **Form** column. **Further information:** "The Form column in Runtime mode", Page 329
	Warning (e.g., if a monitoring section has been removed)
	Error (e.g., 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.

13.6 The Setup mode

Application

Before starting series production, you can check all monitoring sections in the **Setup** mode in the NC program and adapt the settings at one central point.

Description of function



The **Process Monitoring** workspace in the **Setup** mode

The **Process Monitoring** workspace contains the following specific content in the **Setup** mode:

1 The **Setup** area with a table
Further information: "The Setup area", Page 323

2 The **Filter** column for displaying only specific monitoring sections in the table
Further information: "The Filters column in Setup mode", Page 324

3 The **Form** column with settings of the monitoring tasks
Further information: "The Form column in Setup mode", Page 325

13.6.1 The Setup area

Application

In the **Setup** area, the control displays all monitoring sections of the NC program as a table.

When combined with the **Form** column, you can modify parameters, reactions, or entire monitoring tasks quickly and easily.

Description of function

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
- NC block numbers of the monitoring section in square brackets

The following six columns show the selected monitoring tasks.

The **Heat map** column shows if and which monitoring task is displayed in the simulation on the workpiece. Select this monitoring task in the **Heat map** area of the form.

Operation

- Clicking or tapping the **Section** cell at the top left marks the entire table.
- Clicking or tapping a cell in the header marks the entire column.
- Clicking or tapping the first cell of a row marks the entire row.
- Clicking or tapping another cell marks only this cell.
- When only one cell is marked, the control selects the associated NC block in the **Program** workspace.
- If you select an NC block in the **Program** workspace, the control will mark the associated area in the table (e.g., the row of the monitoring section). If the selected NC block is not part of the monitoring section, the control will mark all rows.

The settings in the **Form** column adapt to the current selection.

> If you long-press or right-click in a column, row, or cell, the control will open the **Monitoring task** window for the marked area. The window contains the same table as on the start page with the monitoring tasks as rows and the applications as columns.
>
> You can select new monitoring tasks or remove the monitoring tasks for all marked cells.
>
> **Further information:** "Selecting the monitoring tasks", Page 317

13.6.2 The Filters column in Setup mode

Application

You can filter the table rows in the **Setup** area (e.g., for modifying the settings of all filtered rows at the same time).

Description of function

The control provides the following filtering options:

- **Range selection**

  You can specify line numbers or ranges to be hidden or hide lines using the buttons.
- **Section search**

  The control displays the monitoring sections whose names contain the search term. You can use regular expressions for this purpose.

  Further information: Programming and Testing User's Manual

Using toggle switches, you can specify the filters to be activated. As long as any filters are active, the control displays a filled-in filter symbol in the title bar.

13.6.3 The Form column in Setup mode

Application

The **Form** column contains different content, depending on the selected mode or user interface element.

When the **Setup** mode is active, the form shows settings for the monitoring tasks.

Description of function

The control adjusts the contents of the form to the currently marked cells, rows, or columns in the **Setup** area:

Area	Meaning
Monitoring task	The control displays the use case, the signal, and the method for the marked cells. You can change or remove the monitoring tasks for all marked cells. If you click the **Change** button, the control opens a window that shows the same selection table as on the start page. **Further information:** "Selecting the monitoring tasks", Page 317
Parameter settings	All marked monitoring tasks are displayed as tabs. Each tab displays the parameters of the corresponding monitoring task. You can change the parameterization using sliders. If you marked multiple identical monitoring tasks whose parameterization do not match, the control displays a dot for each parameterization on the slider.
Reactions	You can use the toggle switch to the left of the separator to activate or deactivate whether reactions should be triggered for all marked monitoring sections. To the right of the separator, the control displays all available reactions for one or more marked monitoring tasks. Toggle switches allow you to activate or deactivate the individual reactions. **Further information:** "Reactions", Page 315 If you marked multiple monitoring tasks whose active reactions do not match, the control displays the toggle switch in its center position.
Heat map	You can select one of the defined monitoring tasks whose status is to be displayed on the workpiece in the **Simulation** workspace. **Further information:** Programming and Testing User's Manual
The **Open** and **Save** buttons	You can load a saved monitoring template for the marked row or column or save the current settings as a template.

Except for the **Heat map** setting, the control provides the same options in **Runtime** mode. There, you can use a graph to compare how a parameterization change would affect a recorded machining operation. Each monitoring task must be parameterized separately.

Further information: "Settings for parameterizing of monitoring tasks", Page 335

Note

When removing a monitoring task that is selected as **Heat map**, the control will reset the heat map.

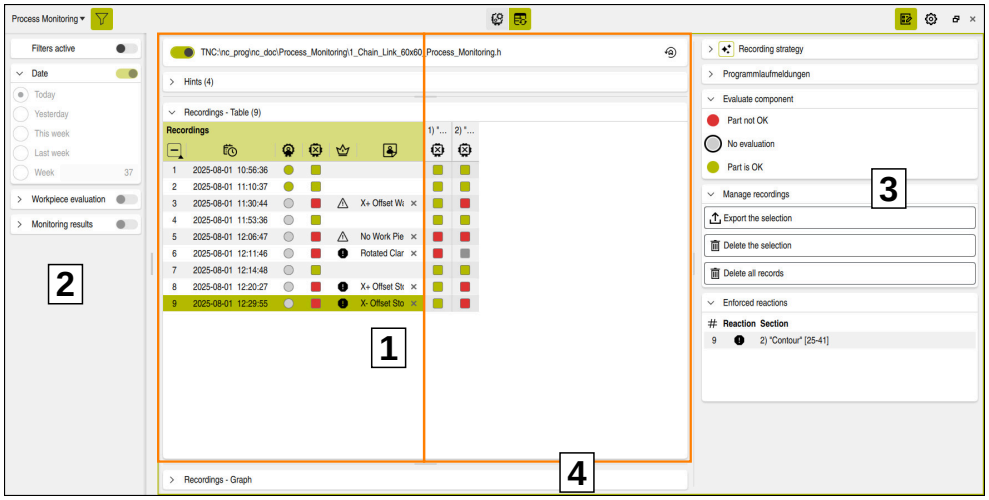
13.7 The Runtime mode

Application

In the **Runtime** mode, the control displays information about the recorded machining operations.

You can view the recordings in detail or compare them with one another. You can also change settings of the recordings and the representation.

Description of function



The **Process Monitoring** workspace in the **Runtime** mode

The **Process Monitoring** workspace contains the following specific content in the **Runtime** mode:

1 The **Recordings - Table** area, sub-divided into a left and a right part

Further information: "The Recordings – Table area", Page 326

2 The **Filter** column for displaying only specific machining operations in the table

Further information: "The Filters column in Runtime mode", Page 329

3 The **Form** column with settings or monitoring tasks, depending on the active content of the **Recordings - Table** area.

Further information: "The Form column in Runtime mode", Page 329

4 The **Recordings - Graph** area for displaying developments of recorded values graphically

Further information: "The Recordings - Graph area", Page 337

13.7.1 The Recordings - Table area

Application

In the **Recordings - Table** area, the control displays all recordings of past machining operations as a table.

You can assess the recordings, view them in detail, and compare them with one another (e.g., in combination with the **Form** column).

Description of function

Next to the area title, the control displays in parentheses how many rows the table contains. If a filter is active, the control shows the number of the currently displayed recordings in the **(x/x)** format.

If no recordings have been made yet, the control will display gray bars as placeholders in the tables.

> Colored boxes represent automatic assessments made by Process Monitoring. Colored circles are assessments you have defined.

Operation

- If you tap or click the right part of the table, the control will enlarge the selected part.
- If you tap or click the icon of a table column, the control will sort the contents of the table by this column in ascending or descending order.
- If you long-press or right-click the icon of the table column, the control will open the **Configure table** window.
- If you tap or click a row once, the control will mark this row in addition to the currently selected row. If you double-tap or double-click a row, the control will mark only this row.

Left part

In the left part, the control displays the **Recordings** table with the recorded machining operations.

The control displays the first three columns 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.
	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 method used, the control will use the evaluated recording as reference machining operations for monitoring, starting with the next monitoring section. 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, the recordings must not include any process disturbances or slower feed-rate values than used while proving-out

The contents of the remaining columns can be selected in the **Configure table** window.

Further information: "The Configure table window", Page 328

Right part

In the right part, the control displays all monitoring sections in the NC program side by side. You can select a monitoring section and view the results of the recorded machining operations as table rows.

The contents of all columns can be selected in the **Configure table** window.

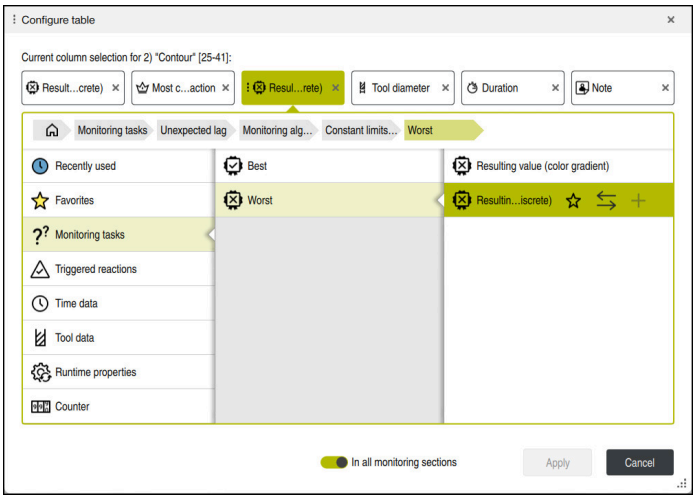
If the **Form** column is active, the control displays the monitoring tasks of the selected monitoring section as a graph.

Further information: "Graphic display of the monitoring tasks", Page 332

The Configure table window

If you long-press or right-click the icon of a table column in the **Recordings - Table** area, the control will open the **Configure table** window. In this window, you can select the information to be displayed in the table for the recordings or the active monitoring section.

The window shows the table columns as tabs aligned next to each other. You can drag the tabs representing the table columns to rearrange the order, or you can delete them.



The **Configure table** window

The **Configure table** window contains the following buttons:

Button	Meaning
☆	**Favorites** You mark your selection as favorite.
⇆	**Replace** The control inserts the selection instead of the current column.
+	**Add** The control inserts the selection as a new column after the current column. The tables can contain up to six columns.
In all monitoring sections	If you activate this toggle switch, the changes will be effective for the tables of all monitoring sections. Applies to the monitoring section tables only
Apply	The control adopts the changes.
Cancel	The control closes the **Configure table** window.

An overview of all selection options can be found here:

Further information: "Overview of the selection options in a table or graph", Page 339

13.7.2 The Filters column in Runtime mode

Application

You can filter the table rows in the **Recordings - Table** area (e.g., for comparing only specific machining operations with one another).

Description of function

The control displays the **Filters** icon only when the table has content.

The control provides the following filters:

- **Date**
  Day or week of machining
- **Workpiece evaluation**
  User's assessment
- **Monitoring results**
  Process Monitoring active or inactive, with or without process disturbance

Using toggle switches, you can specify the filters to be activated. As long as any filters are active, the control displays a filled-in filter symbol in the title bar.

13.7.3 The Form column in Runtime mode

Application

The **Form** column contains different content, depending on the selected mode or user interface element.

Description of function

When the **Runtime** mode is active, the form shows the following contents, depending on the active element.

- When the left part of the **Recordings - Table** area is active, the form shows settings regarding the recordings.
- When the right part of the **Recordings - Table** area is active, the form shows the monitoring tasks of the marked recordings as a graph.

Recording settings

If the left part of the table is active in the **Recordings - Table** area, the control will display the following settings in the **Form** column:

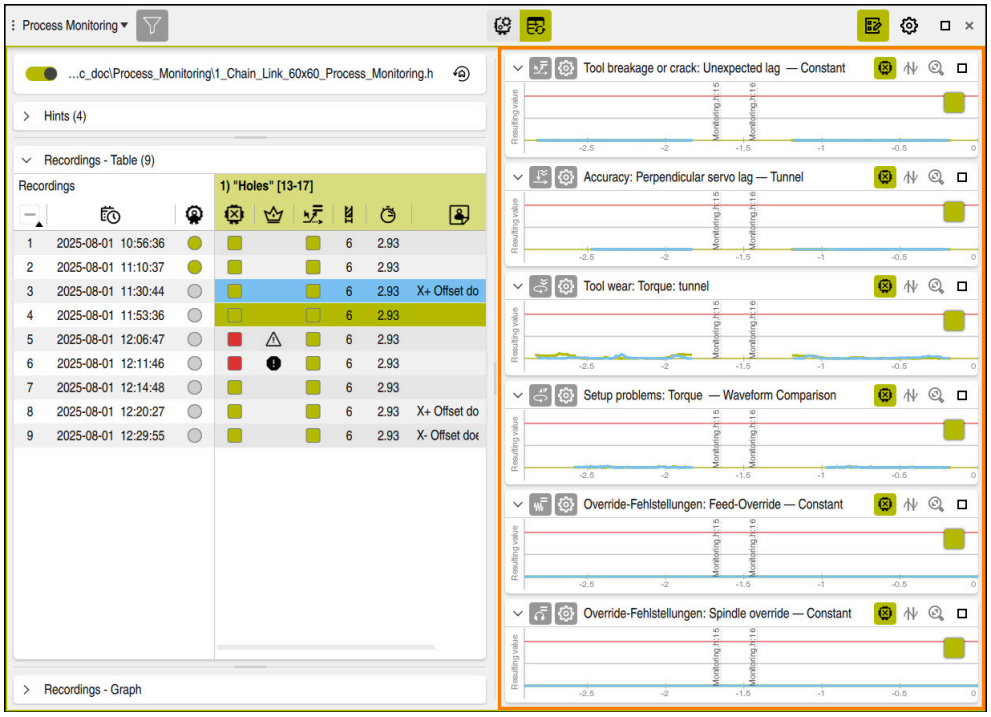
Area	Meaning
Recording strategy	You can select the following recording strategies: ■ **Record each operation completely** The control records the entire information of all machining operations. ■ **Limit: Record up to n operations** You define the maximum number of machining operations to be recorded. ■ **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. Once process monitoring has been set up completely, you can use this strategy for monitoring and logging, but not for reference machining operations. This strategy significantly reduces the amount of data. ■ **Interval: Record each nth operation** You can specify after which number of machining operations the control will record process data. For the other machining operations, only meta-information will be recorded. ■ **Interval: Record each nth operation and critical operations** This strategy has the same effect as the previous one. 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. Once the maximum disk space is used up or the limit you have set is reached, the control will overwrite recordings using one of the following strategies: ■ **Newest recording** ■ **Oldest not-evaluated recording** ■ **Never** The first recording will never be overwritten. If the control is not allowed or able to overwrite previous recordings, the control will interrupt process monitoring and display an error message. If the current settings have optimizing potential (for example, because they use up much disk space), the control will display a flashing icon in the title bar of this area.  If you select this icon, a window opens where an optimization suggestion is displayed. You can either accept or ignore the suggestion.

Area	Meaning
Program-run notifications	Using toggle switches, you can specify whether the control will show or suppress the following warning messages for this NC program: ■ **Insufficient pre-positioning** In case the tool position at the beginning of the monitoring section deviates too far from the first recording ■ **Setting for path calculation was changed** Indicates a change in path calculation that makes the current machining operation unsuitable for comparison with the reference machining operation. ■ **Rapid-traverse override has changed** ■ **Inactive monitoring sections** In case process monitoring becomes inactive during program run
Evaluate component	You can evaluate all selected machining operations as **Part is OK** or **Part not OK**. Depending on the method used, the control will use the evaluated recording as reference machining operations for monitoring, starting with the next monitoring section. 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 **Recordings** table. ⓘ You can evaluate only completely executed recordings. Good parts must be representative for the machining process. For example, the recordings must not include any process disturbances or slower feed-rate values than used while proving-out
Manage recordings	■ **Export the selection** You can save the meta-information of the selected machining operations as HTML, PDF, CSV, or JSON files. ■ **Delete the selection** The control deletes all selected recordings. You cannot delete the first recording in this way. ■ **Delete all records** The control deletes all recordings, including the first one. ⓘ 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.

Area	Meaning
Triggered reactions	The control displays the triggered reactions for all rows marked in the **Recordings - Table** area. The control displays the following information: ■ Recording number ■ Reaction icon ■ Name of the monitoring section When you select a reaction, the control will open the associated recording in the right part of the table.

Graphic display of the monitoring tasks

If the right part of the table in the **Recordings - Table** area is active, the control will display the graphs for the monitoring tasks in the **Form** column. The control displays the values of all marked recordings.

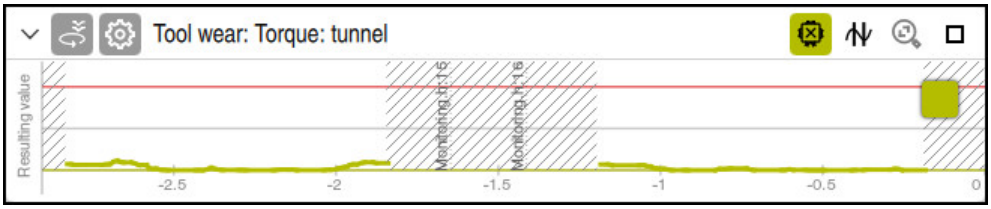


The **Form** column with the graphs of the monitoring tasks

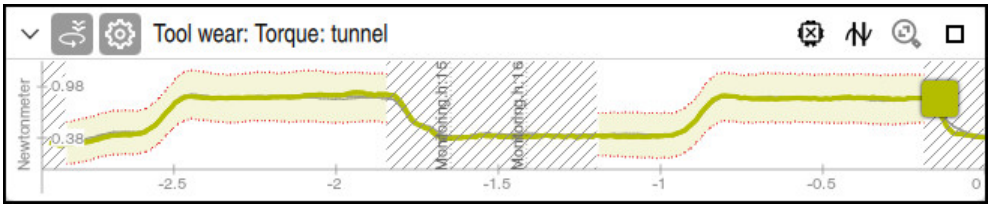
During program run, the control always displays the monitoring tasks with their current values. The time axis is divided into seconds, or into minutes for longer monitoring sections.

The graphs represent the machining operation signals. If the control is unable to evaluate the monitoring task, it will display the corresponding graph area as dots or hatched. If you tap or click this area, the control will open 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 will open the **Monitoring task** window that shows the same table as on the start page. You can change or remove a monitoring task. **Further information:** "Selecting the monitoring tasks", Page 317
	Open or close the **Settings** for parameterization **Further information:** "Settings for parameterizing of monitoring tasks", Page 335
	Changing the **Signal display** 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 records. 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.
	Show or hide the **Intersecting lines** If you select this icon, the control will display two vertical lines in the graph. You can move these lines. The control will mark the intersections of the graphs with the lines. Next to the intersections, the control displays the resulting value color or the numerical values of the signal, depending on the active **Signal display**. While the signal run is active, the control will display the points of intersection with the signal and all limit values. If multiple recordings have been marked, the symbol is dimmed.

Icon	Meaning
	Reset scaling Display the graphs of the entire monitoring section If the icon is dimmed, the control displays the entire graph.
	At the right edge of the graph, the control will display the worst resulting value of the entire monitoring section, independent of the visible part of the graph. Up to half the value of the error limit, the display is green; from the half onwards yellow; and from the error limit on, it is red.

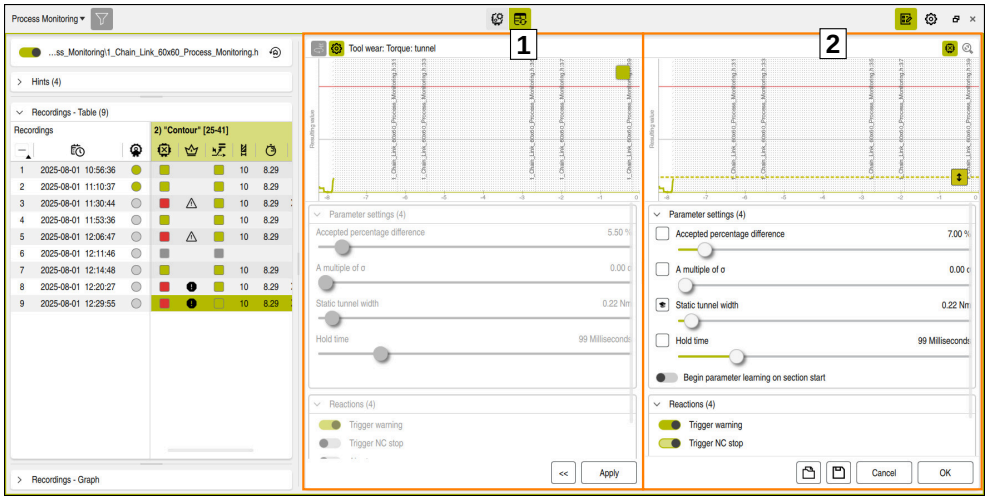
If a graph has been collapsed, the control shows only the symbols for the monitoring task and the worst resulting value.

Using the graphs

- By scrolling or dragging the graph, you can enlarge or reduce its size horizontally.
- You can shift the graph by swiping or by dragging 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-tap or double-click a position within the graph, the control will select the corresponding NC block in the NC program and in the graph.
- If you select an NC block within the **Program** workspace, the control will mark the corresponding NC block in the graph.

Settings for parameterizing of monitoring tasks

The **Settings** icon within the monitoring tasks allows changing the parameterization of the monitoring task individually.



Parameterization of a monitoring task

Within the settings of a monitoring task, the control offers two areas:

1 Parameterization of marked recording

The control dims the parameterization that was active at the time of the marked recording. This area is collapsed by default.

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.

2 Preview of current parameterization

The control displays the current parameterization for the monitoring task. When changing the settings, the control displays which result this recording would have achieved with the new parameterization.

When the complete graph is visible, the control displays the worst resulting value in the slider icon.

The settings of monitoring tasks contain the icons and buttons below:

Icon or button	Meaning
	Parameter learning This icon marks the parameter you can modify with the slider in the graph. **Further information:** "Parameter learning", Page 344
	Slider The dashed line marks the worst resulting value of the machining operation. The slider defines the distance of this deviation from the error limit. The slider shows the color which this machining operation would have achieved with the new parameters. Only for the **Resulting value** signal representation

Icon or button	Meaning
Begin parameter learning when monitoring section starts	When activating the toggle switch, the control will automatically start the learning phase for the marked parameter in the next machining operation. When parameter learning is successful, the control will deactivate the toggle switch again. **Further information:** "Learning the parameter in the first machining operation", Page 346
>>	**Expand**
<<	**Collapse** In the area of the marked recording
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.
	**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.

Apart from parameter learning, in the **Form** column of the **Setup** mode, the control offers the same options. You can parametrize several monitoring tasks simultaneously there, but you cannot compare the changes with the previous parameterization.

Further information: "The Form column in Setup mode", Page 325

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).

13.7.4 The Settings window

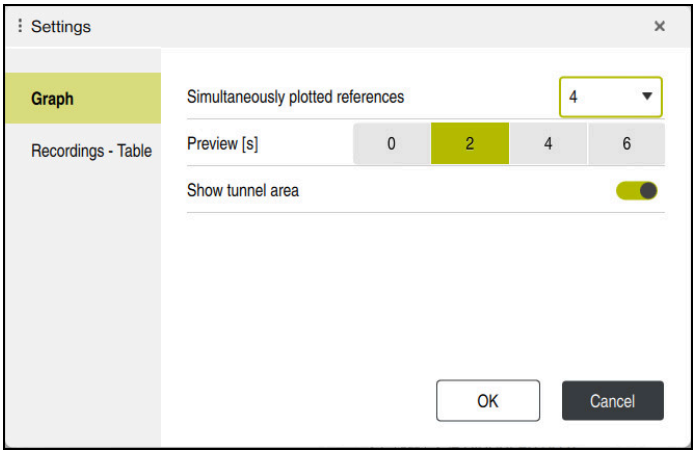
Application

In the **Settings** window, the control provides settings for the monitoring task graphs and the **Recordings - Table** area.

Description of function

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 **2...10** that the control displays simultaneously as graphs in the monitoring tasks.
Preview [s]	During execution, the control displays graphs of the current monitoring tasks. You can show an area at the right end of the graphs where the control displays the expected signal run, based on the reference machining operations. Select how many seconds **0...6** the control will preview
Show tunnel area	When the toggle switch is active, the control displays the monitoring tunnel area in the graph on a color background. Only for monitoring tasks with the **Tunnel** or **Constant** method.

The Recordings - Table area

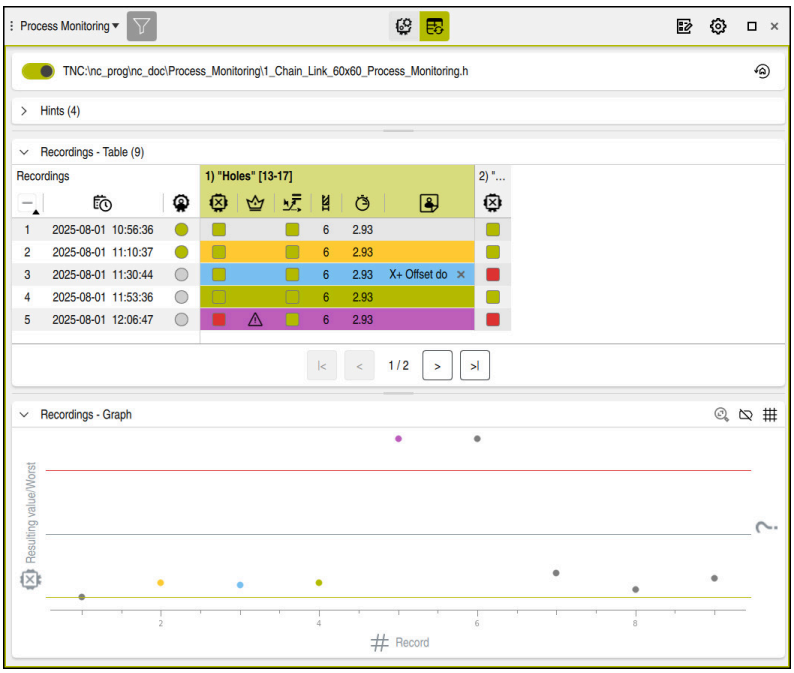
In the **Recordings - Table** area, you can define the number of decimal places **0...8** for the display of floating-point numbers in the table.

13.7.5 The Recordings - Graph area

Application

In the **Recordings - Graph** area, the control by default shows the history of the resulting value across the existing recordings. You can change the displayed values (for example, in order to detect correlations or trends).

Description of function



The **Process Monitoring** workspace with the **Recordings - Graph** area opened

You can configure the graph axes separately for all recordings or single monitoring sections, depending on the active part of the **Recordings - Table** area. After restarting the control, the default values are active again.

You can choose any values to be shown by the individual axes of the graph. As an option, you can also select a second Y axis on the right and thus represent two values relating to one common X axis.

If you tap or click the gray description of an axis, the control opens the **Configuration of graph and axes** window. This window allows you to select the value to be represented by the axis.

Further information: "Overview of the selection options in a table or graph", Page 339

The recording graph contains the following icons:

Icon	Meaning
	Reset scaling If the icon is dimmed, the control displays the entire graph.
	Show or hide the **Axis label**
	Show or hide the **Grid**
	Circles show the value of the left Y axis. When rows are marked in the **Recordings - Table** area, the circles are displayed in the respective color.
	Triangles show the value of the optional right Y axis. When rows are marked in the **Recordings - Table** area, the triangles are displayed in the respective color.

13.7.6 Overview of the selection options in a table or graph

The following table contains an overview of the data you can select during process monitoring at the following locations:

- Table columns in the **Recordings - Table** area
  Selection in the **Configure table** window
  Further information: "The Configure table window", Page 328
- Axes in the **Recordings - Graph** area
  Selection in the **Configuration of graph and axes** window
  Further information: "The Recordings - Graph area", Page 337

Depending on whether the left or right part is active in the **Recordings - Table** area, the control will display different options for selection.

Icon	Group	Icon	Selection option
??	**Monitoring tasks**		For each signal, select one of the two variants: ■ **Monitoring algorithm** and colored display **Further information:** "Display mode", Page 342 ■ Numerical **Statistics** for this signal **Further information:** "Statistics", Page 343 **Further information:** "Monitoring tasks", Page 309
	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 triggered reaction **Further information:** "Reactions", Page 315
	Time data		**Recording**
			Date
			Time Time at the start of the NC program or monitoring section
			Date and time Date and time at the start of the NC program or monitoring section
			Duration Duration of the NC program or monitoring section in seconds
	Workpiece evaluation		**Manual evaluation**
	Tool data		**Tool name**

Icon	Group	Icon	Selection option
			Tool diameter
			Delta radius DR (tool table)
			Delta radius DR (NC program)
			Tool length
			Delta length DL (tool table)
			Delta length DL (NC program)
			Tool number
			Tool index
			Number of edges
			Tool life
			Maximum tool life
			Cutting material
			ID for central tool management
			Comment
			Tool type
	Runtime properties		**Resulting value** Result of the entire machining operation You select the display mode. **Further information:** "Display mode", Page 342
			Note Only in the **Configure table** window You can enter a comment.
			Contains process data Check box You can only select recordings as reference machining operations that contain process data.
			Pallet name Only in the **Configure table** window Only if the NC program is part of pallet machining or pallet handling
		MB	**Memory requirements** File size of this recording in MB

Icon	Group	Icon	Selection option
			Possible teach-in cuts for reference Only in the **Configure table** window Number of possible reference machining operations for this monitoring section When you select the number, the control will display the reference machining operations.
		32	**Fingerprint of path calculation** Verification number The control considers the functions and settings that influence path calculation, such as Cycle **32 TOLERANCE**. If the control displays a different number, the path calculation settings have changed. This machining operation is possibly no longer comparable with the previous machining operations.
	Counter **FUNCTION COUNT** NC function		**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

Display mode

For monitoring tasks and the **Resulting value**, you can specify how the control will display the result:

Display mode	Icon	Selection option
Worst or **Best**		**Best** The control displays the smallest resulting value of the machining operation (i.e., the best result). By selecting this option, you can see, for example, whether the machining operation within a monitoring section ever matched the reference operation.
		Worst The control displays the largest resulting value of the machining operation (i.e., the worst result). Using this option, you can see at a glance whether the machining operation, for example, exceeded half the tolerance or whether a process disturbance occurred. By selecting Resulting quantity (color gradient), you can see how close the machining operation was to the error limit.
Resulting quantity (discrete) or **Resulting quantity (color gradient)** Only in the **Configure table** window	No icon	**Resulting quantity (color gradient)** The control displays the result as a color transition from green to dark red. With this option, you can, for example, see whether a machining operation was approaching the error limit or exceeded it greatly.
	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: Machining was in the range between half of the tolerance and the error limit ■ Red: Error limit exceeded ■ Gray: Program run aborted Using this option, you can see at a glance whether the error limits were exceeded.

If no result is available yet, the control displays a gray icon in the **Recordings - Table** area.

Statistics

Instead of the color-based display of the method, you can also use statistics to display numerical values related to the monitoring task signals:

Icon	Selection option
	Minimum The control displays the smallest value of this signal during the machining operation.
	Maximum The control displays the largest value of this signal during the machining operation.
	Average The control displays the mean value of this signal during the machining operation.
	Peak-to-peak The control displays the difference between the smallest and the largest value of this signal during the machining operation.
	Standard deviation The control displays the standard deviation σ of this signal during the machining operation. The less the signal varies from the average value, the smaller the standard deviation.

If no result is available yet, the control displays a gray icon in the **Recordings - Table** area.

13.8 Parameter learning

Application

The parameter learning function allows you to set the Process Monitoring parameters without knowing the required values. You can define the maximum deviation of the machining operation from the error limit. The control will use this information to calculate the parameter values.

This function can also be used during machining.

If you use a monitoring task with the **Constant** method, you can monitor even the first machining operation based on a parameterization matching the workpiece.

Related topics

- Changing the parameterization while not machining
  Further information: "Settings for parameterizing of monitoring tasks", Page 335
- Changing parameter values in **Setup** mode
  Further information: "The Form column in Setup mode", Page 325

Requirements

- The **Constant**, **Tunnel** or **Waveform comparison** methods
- The **Resulting value** signal representation

Description of function

Parameter learning is based on the **Resulting value** signal representation in the graph of the monitoring task.

In the **Resulting value** representation, the control shows the signal being evaluated regarding the error limits. When the signal approaches the red line, machining deviates from the records.

The control shows a dashed line at the height of the largest deviation during the machining operation. You can shift this line up and down and thus define the distance between the largest deviation in the machining operation and the error limit. The control calculates the value of the marked parameter to achieve the defined result.

The closer the dashed line is to the error limit, the more sensitively process monitoring reacts to deviations (e.g., with a smaller tunnel width).

The control provides parameter learning at the following points in process monitoring:

- In the settings for parameterization in the **Runtime** mode
  The control uses data from existing recordings.
  Further information: "Settings for parameterizing of monitoring tasks", Page 335
- During machining
  The control uses the machining data.
  Further information: "Parameter learning during machining", Page 345

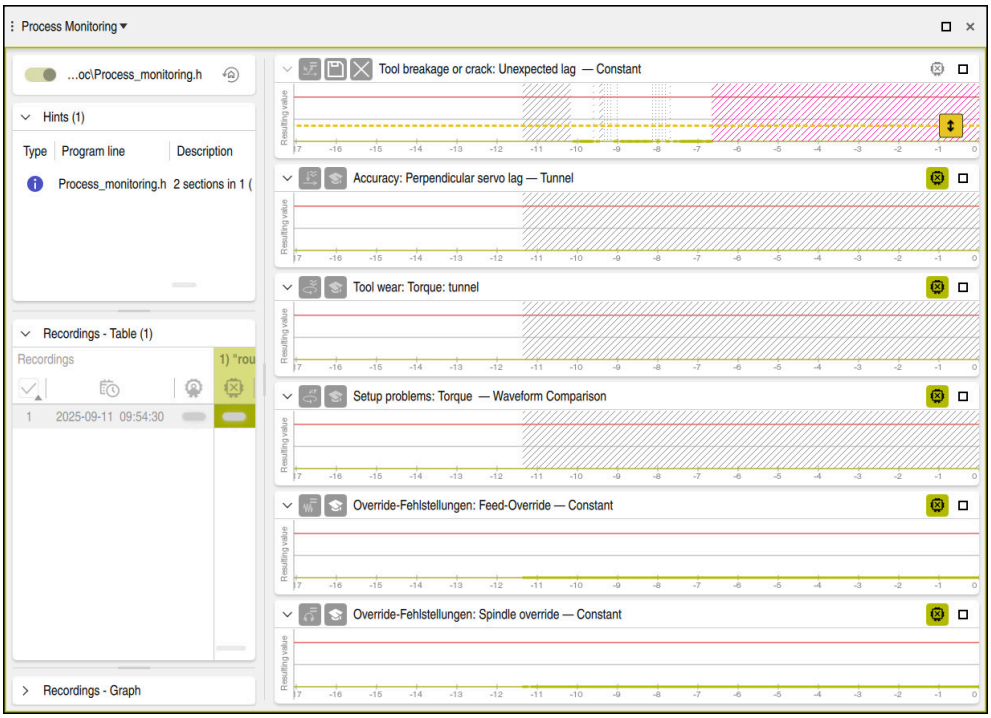
Parameter learning during machining

During program run, you can use parameter learning for parameterization with the data of the current machining operation.

To obtain meaningful values, the control must learn the parameter in the heaviest machining operation of the monitoring section (e.g., roughing operation).

In case of monitoring tasks with the **Constant** method, you can use parameter learning in the first machining operation. This allows you to monitor even the first machining operation with parameter values suited for the workpiece.

The **Tunnel** and **Waveform comparison** methods require at least one reference machining operation.



Machining with active learning phase

During operation, the control displays the following icons for learning parameters:

Icon	Meaning
	Start **Learning phase** The control no longer performs monitoring, but instead collects data for calculating the parameter. During the learning phase, the control displays the monitoring task graph as a hatched pink graph.
	Slider The dashed line marks the worst resulting value of the machining operation. The slider defines the distance of this deviation from the error limit. The slider shows the color that this machining operation will achieve with the learned parameters.
	Save The control calculates the parameter values to suit the slider position and monitors with the new values.

Icon	Meaning
	If the control is not able to calculate the value, it will cancel the leaning phase and the monitoring for the current monitoring section. The control displays a message that it was unable to calculate the parameter.
	**Cancel** the learning phase The control displays a safety prompt asking if you wish to stop monitoring up to the end of the monitoring section or continue with the initial values.

If you do not save the learning phase before the end of the monitoring section or cancel it, the control keeps the initial parameter value.

13.8.1 Learning the parameter in the first machining operation

In the example below, the control learns the parameter of the monitoring task **Unexpected servo lag — Constant** at the beginning of the first machining operation.

Preconditions:

- Monitoring sections have been defined with **MONITORING SECTION**
- Heaviest machining operation at the beginning of the monitoring section

To obtain meaningful values, the control must learn the parameter in the heaviest machining operation of the monitoring section (e.g., roughing operation). If the heaviest machining operation does not occur at the beginning of the monitoring section, start the learning phase manually.

The following steps are required:

1 Activating process monitoring
2 Activating parameter learning at the start of machining
3 Setting the value during machining

Activating process monitoring

To activate process monitoring:

- ▶ Open the NC program in the **Program Run** operating mode
- ▶ Open the **Process Monitoring** workspace
- > The control displays the process monitoring start page.
- > The control marks a pre-selection of monitoring tasks.

- ▶ Select **Activate**
- > The control activates process monitoring with the selected monitoring tasks for this NC program.

Activating parameter learning at the start of machining

To select a parameter for learning during the first machining:

- ▶ Activate the **Runtime** mode
- ▶ If applicable, open the **Form** column
- ▶ Select the right part of the table in the **Recordings - Table** area
- ▶ Select the desired monitoring section as needed
- > The control displays an area for each monitoring task in the form
- ▶ Open **Settings** for parameterization within the desired monitoring task
- > The control displays the **Parameter settings** and **Reactions** areas.
- > The **Upper limit** parameter is marked for parameter learning.
- ▶ Activate the **Begin parameter learning when monitoring section starts** toggle switch
- ▶ Select **OK**
- > The control saves the parameter settings.

Setting the value during machining

To set the value during machining:

- ▶ Run the NC program
- > As soon as the monitoring section starts, the control automatically starts the learning phase and does not perform monitoring.
- > The control displays the monitoring task graph as a hatched pink graph.
- ▶ Drag the slider to the desired position at the error limit

> The dashed line marks the worst resulting value of the machining operation. The closer the icon is to the red line, the more sensitively process monitoring reacts to deviations.

- > The slider shows the color that this machining operation would achieve with the new parameters.
- ▶ Select **Save**
- > The control calculates the parameter values to suit the slider position and monitors with the new values.

> If the control is not able to calculate the value, it will cancel the leaning phase and the monitoring. The control displays a message that it was unable to calculate the parameter.
>
> In this case, you have the following options:
>
> - Restart the learning phase with the icon
> - Longer learning phase
> - Select another parameter

Notes

- In case of monitoring tasks with the **Tunnel** or **Waveform comparison** method, you cannot learn during the first machining operation. These methods always require reference machining operations.
- When you select a pink hatched area in the graph within the recording, the control will display the result of the learning phase. If the learning phase was successful, the control displays a new value of the parameter.
- You may start the learning phase for a parameter several times during program run. When you select the pink hatched area in the graph, the control will display the value after the last successful learning phase.

14

CAD Viewer

14.1 Fundamentals

Application

CAD Viewer allows you to open CAD files on the control (e.g., to set a workpiece preset in the model and load it to an NC program).

The CAD Import (#42 / #1-03-1) function allows you to save contours and positions from a CAD file as Klartext programs or point files.

The **3D mesh** (#152 / #1-04-1) function allows you to rectify errors in 3D models.

Related topics

- Importing contours and positions with CAD Import from **CAD Viewer** (#42 / #1-03-1)

  Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 362
- Rectifying errors in 3D models with **3D mesh** (#152 / #1-04-1)

  Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 370
- Creating 2D sketches on the control

  Further information: Programming and Testing User's Manual

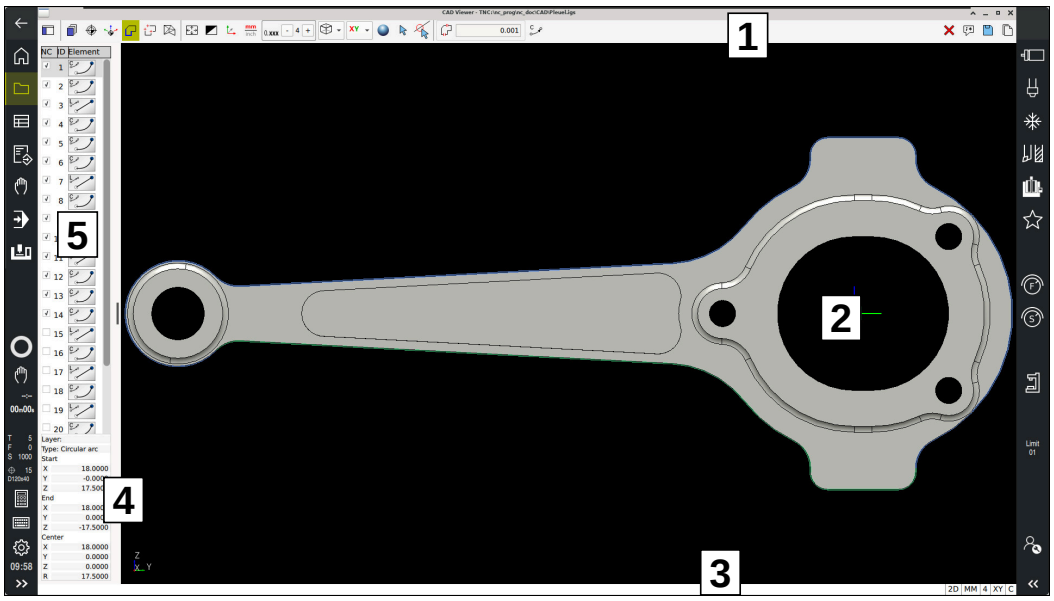
Description of function

CAD Viewer runs as a separate application on the third desktop of the control.

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

Screen layout



CAD file open in **CAD Viewer**

CAD Viewer contains the following areas:

1 Menu bar
Further information: "Menu bar icons", Page 352

2 Graphics area
In the Graphics area, the control displays the CAD model.

3 Status bar
In the status bar, the control displays the active settings.

4 Element information area
Further information: "The Element Information area", Page 354

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 and Element Information areas
	**Display the layer** Display the layer(s) in the List View area **Further information:** "Layer", Page 355
  	**Preset** Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD file", Page 357
 	**Datum** Set the datum Datum has been set **Further information:** "Workpiece datum in the CAD file", Page 360
	**Contour** Select contour (#42 / #1-03-1) **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 362
	**Positions** Select positions (#42 / #1-03-1) **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 362
	**3D mesh** Create a 3D mesh (#152 / #1-04-1) **Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 370
	**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** always calculates internally with mm. If you select inches as the unit of measure, **CAD Viewer** will convert all values to inches. **Further information:** "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 362

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 363 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 362
	Toggle a 3D model between a solid model and a wire-frame model.
	**Select** contour elements (e.g., to obtain element information) **Further information:** "The Element Information area", Page 354
	**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 362
	Activate or deactivate **Hover** If you swipe across the CAD model during touch operation, the controls display selectable options in orange. The function is comparable to hovering with a mouse. As soon as you stop the movement, the control will mark the current selection option. If this icon is active, you can no longer rotate the CAD model.
	**Undo an action**
	**Delete entire list**

Icon	Meaning
	Setting for whether comments are written to NC output files. Add 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 366
FMAX M99	**H file with FMAX M99 at each position (for older NC controls)** Add each position with rapid traverse **FMAX** and cycle call **M99** to the NC program (#42 / #1-03-1) The control displays the icon if you select the **Positions** mode. **Further information:** "Applying positions", Page 364
	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.

The 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.

Manipulating the graphics in CAD Viewer

You can use the mouse or touch gestures to perform the following functions on the CAD model:

Function	Mouse operation	Touch operation
Show the selection options on the CAD model in orange	Hover over the element with the mouse pointer	Swipe across the elements The **Hover** mode is active  **i** As soon as you stop the movement, the control will mark the current selection option.
Zooming Magnify or reduce the CAD model	Scroll with the mouse wheel	Spread or pinch 
Move the CAD model	Drag with the right mouse button held down	Two-finger drag 
Reset the CAD model to its original size	Double-click with the right mouse button	Double tap 
Reset the CAD model to its original size and angle The **Add** mode 	Double-click with the right mouse button Press the **SHIFT** key in other modes	Double tap 
Rotate the CAD model The **Hover** mode is inactive 	Drag with the left mouse button held down	Swipe 
For the selection of positions: Drag a box around the desired area for multiple selection The **Add** or **Remove** mode 	Drag with the left mouse button held down Press the **SHIFT** or **CTRL** key in other modes	Swipe 

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

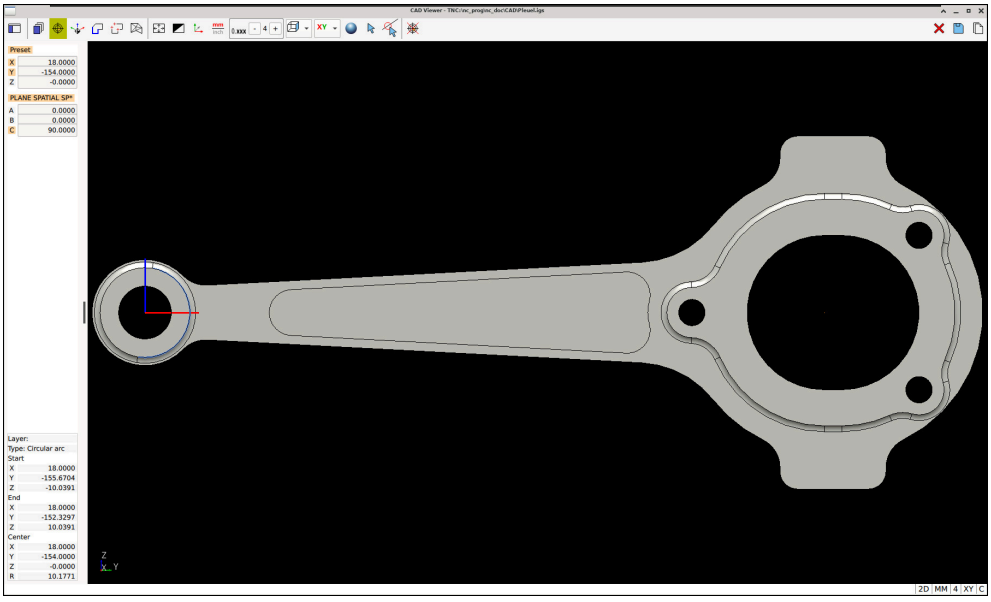
  Further information: "Presets in the machine", Page 182

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 input of numerical values into 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 ;origin = X... Y... Z...
5 ;origin_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 also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.
> - The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.
>
> **Further information:** "Manipulating the graphics in CAD Viewer", Page 356

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:

- ▶ Select **Preset**
- ▶ Position the cursor on the desired element
- > If you are using a mouse, the control displays selectable presets for the element using gray icons.
- ▶ Click the icon at the desired position
- > The control sets the workpiece preset to the selected position. The control turns the icon green.
- ▶ Orient the working plane, if required

Setting the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at the intersection of lines, full circles, and arcs.

To set the workpiece preset at the intersection of two elements:

- Select **Preset**
- Click the first element
- The control highlights the element in color.
- Click 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 182

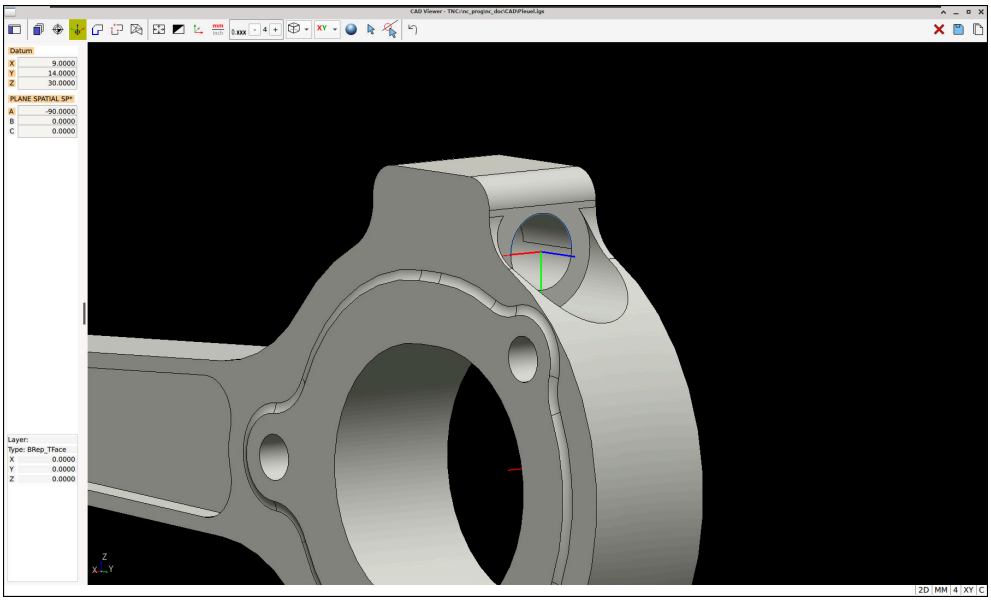
Description of function

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 357

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 358

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+20 Y+25 Z-5
5 PLANE SPATIAL SPA+0 SPB+0 SPC+20 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+20 Y+25 Z-5
5 ;PLANE SPATIAL SPA+0 SPB+0 SPC+20 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

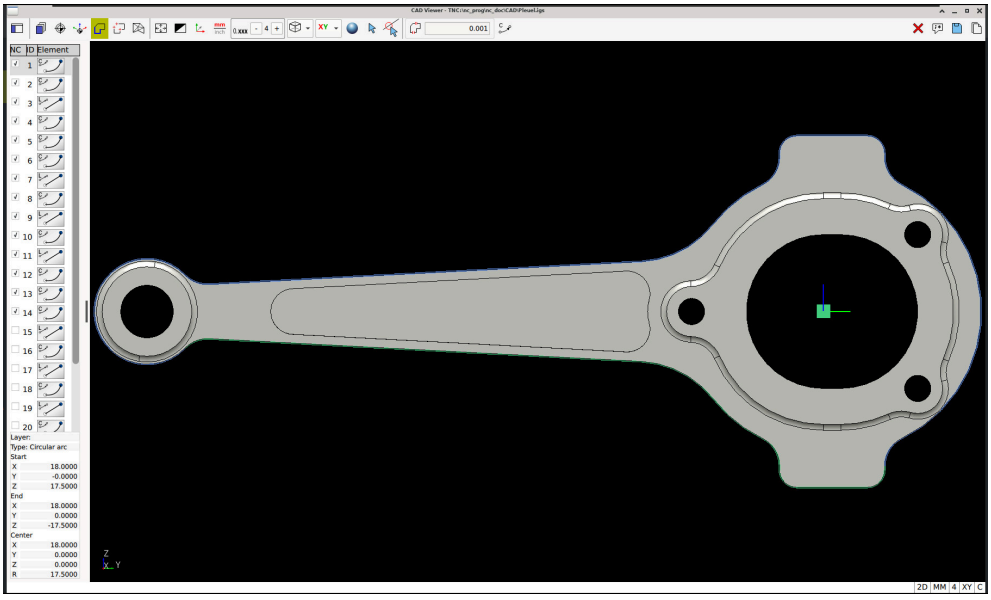
Requirement

- 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 700



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 351

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. All contour elements below the rotary axis cannot be selected and are highlighted 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: "Select positions", Page 367

The following file types are available:

- Point table (**.PNT**)
- Klartext program (**.H**)

If you save the positions to a Klartext program, the control creates a straight line **L** for every position. The output depends on the icon **H file with FMAX M99 at each position (for older NC controls)**.

Example: active icon

```
11 L X+5 Y+5 Z+0 FMAX M99
```

Example: inactive icon

```
11 L X+5 Y+5 Z+0
```

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 all steps with touch gestures.
  Further information: "Manipulating the graphics in CAD Viewer", Page 356
- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.

Selecting and saving 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 press the **CTRL** key and simultaneously select an element, the control deselects the element for export.

Selecting paths independently of existing contour elements

To select a path independently 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 Select positions

- The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.
  Further information: "Manipulating the graphics in CAD Viewer", Page 356
- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.
  Further information: "Selecting and saving a contour", Page 366

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.

FMAX
M99

- Insert **FMAX** and **M99** in the NC program if necessary

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.

FMAX
M99

- Insert **FMAX** and **M99** in the NC program if necessary

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.

FMAX
M99

- Insert **FMAX** and **M99** in the NC program if necessary

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 with that used in the **CAD Viewer**. Elements that have been copied from the **CAD Viewer** to the clipboard do not contain any information about the unit of measure.
- The control 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
- Exporting the simulated workpiece as an STL file
- Using an STL file as workpiece blank
  Further information: Programming and Testing User's Manual

Requirement

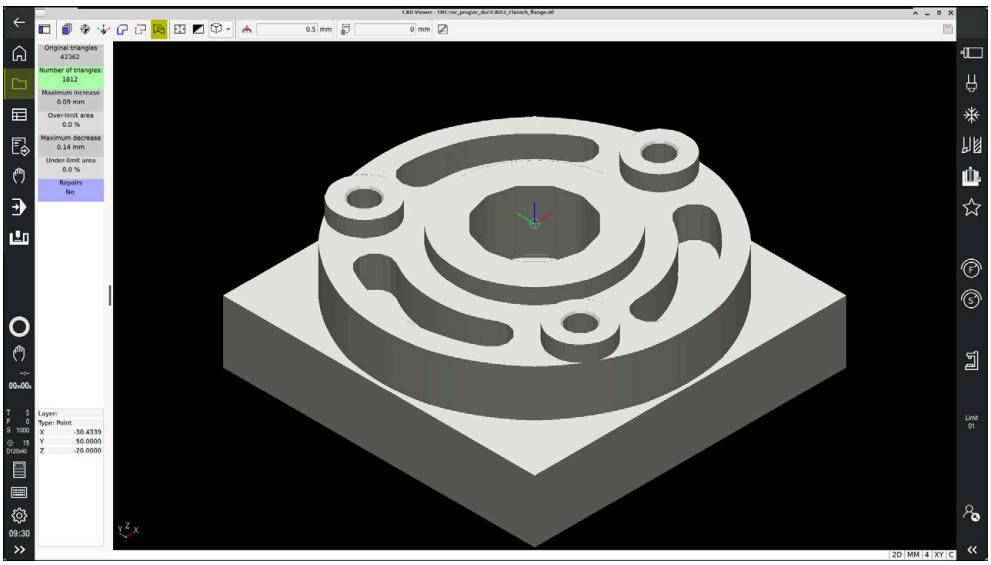
- Software option CAD Model Optimizer (#152 / #1-04-1)

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
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 372
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.
Time	Duration of simplification

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.

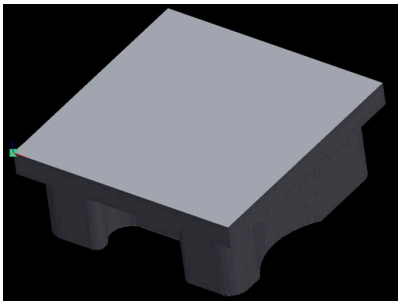
14.5.1 Positioning the 3D model for rear-face machining

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 372

- 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.

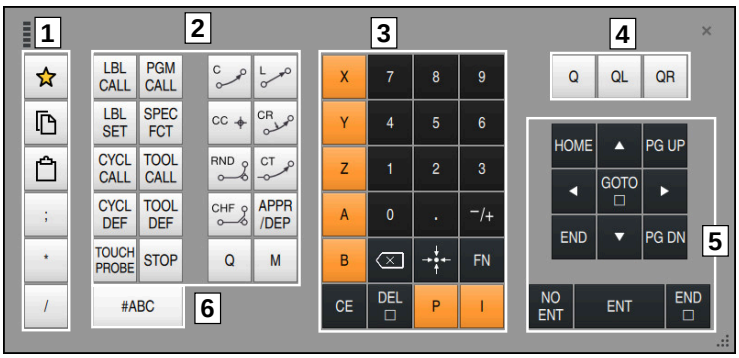
You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

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 NC functions
3 Axis keys and numerical input
4 Q parameters
5 Navigation and dialog keys
6 Switch to text input

If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
- **Q**
- **QL**
- **QR**

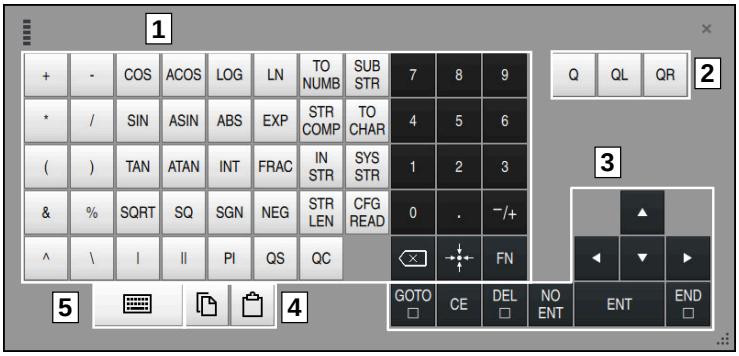
Text input areas

Virtual keyboard in text input mode

Text input mode contains the following areas:

1 Input
2 Navigation and dialog keys
3 Copying and pasting
4 Switch to formula input

Formula input areas



Virtual keyboard in formula input mode

Formula input mode contains the following areas:

1 Input
2 Q parameters
3 Navigation and dialog keys
4 Copying and pasting
5 Switch to NC 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:

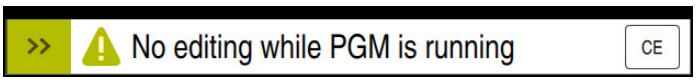
Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot clear this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has 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

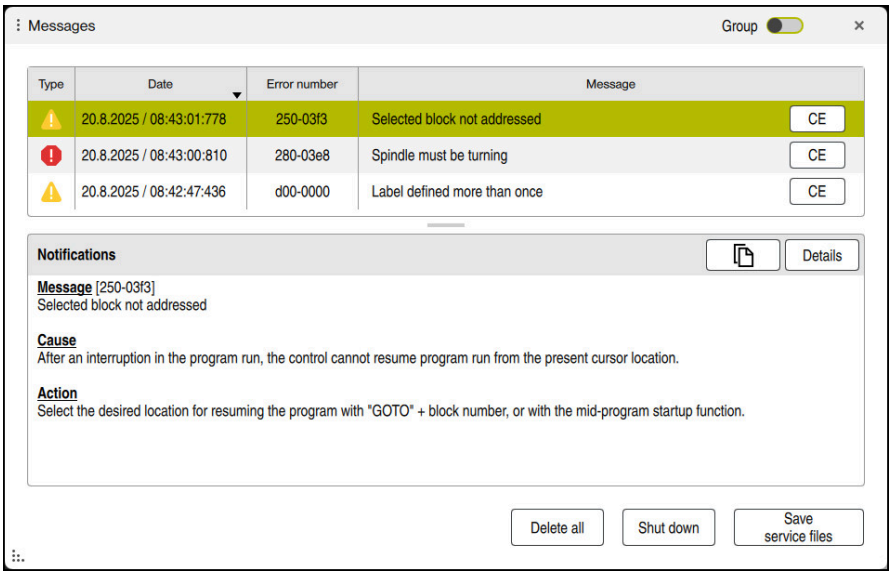
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

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
- Date
- Error number
- Message
- Additional information (root cause, correction, information on the NC program)
  You can copy additional information to the clipboard with a button.

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 381
- 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 382

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.

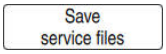
The file name of each service file consists of a user-defined name and a timestamp.

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

- ▶ Select **OK**
- > The control saves the service file in the **TNC:\service** folder.

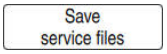
Using a toggle switch, you can define whether the control will save data from Process Monitoring (#168 / #5-01-1) for the current NC program in the service file.

15.2.2 Creating a service file automatically

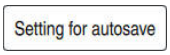
You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

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
- Select the **Active** checkbox
- > 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.

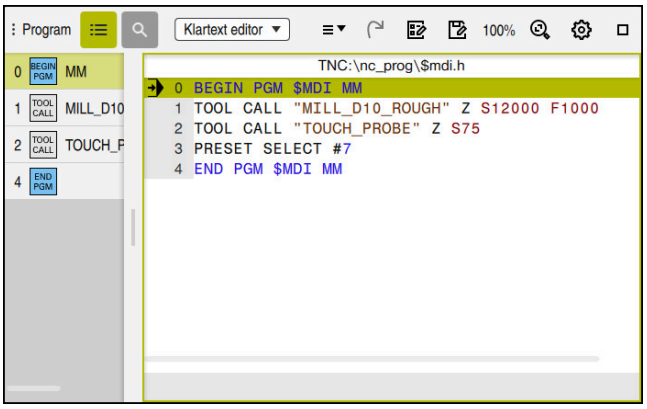
You can also create an NC program step by step. The control remembers modally effective program information.

Related topics

- Creating NC programs
  Further information: Programming and Testing User's Manual
- Running NC programs
  Further information: "Program run", Page 425

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:

- **Document**
  Further information: Programming and Testing User's Manual
- **GPS** (#44 / #1-06-1)
  Further information: "Global Program Settings (GPS) (#44 / #1-06-1)", Page 291
- **Help**
  Further information: Programming and Testing User's Manual
- **Positions**
  Further information: "The Positions workspace", Page 127
- **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 135
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 376

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.
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 431
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 290
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 203
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 432
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, program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 432

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 520
- Datum table
  Further information: Programming and Testing User's Manual
- Reference systems
  Further information: "Reference systems", Page 216
- Preassigned variables
  Further information: Programming and Testing User's Manual

Requirements

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 404

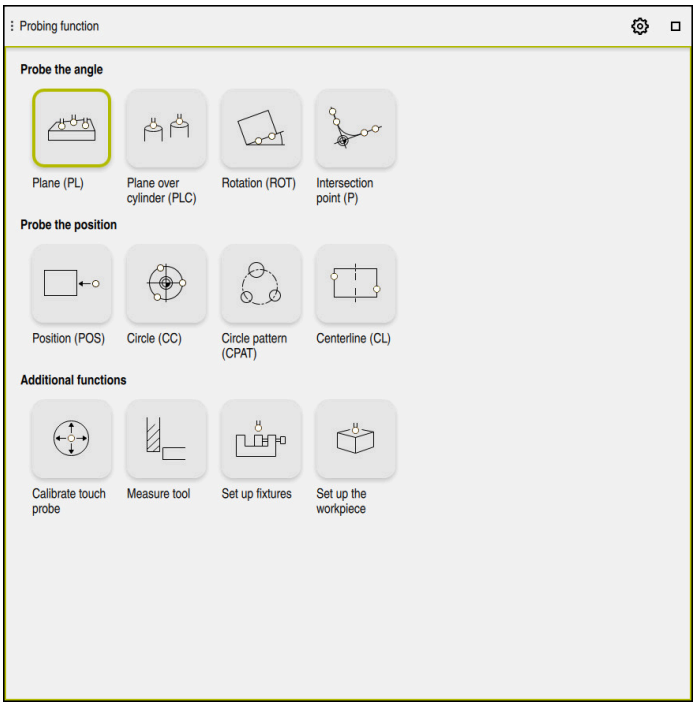
Description of function

The control provides the following functions for setting up the machine in the **Manual** operating mode in the **Setup** application:

- Setting a 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: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258
- **Set up the workpiece** (#159 / #1-07-1)

  Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 411



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 and align the plane, if necessary.
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 and align the plane, if necessary.
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 for the basic rotation of a workpiece", Page 401
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.

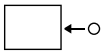
Further information: "Preset table *.pr", Page 520

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 422

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 398
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:** "Setting the circle center point of a stud by means of the automatic probing method ", Page 399
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 404
Measure tool 	The **Measure tool** function allows you to measure tools by touch-off. In this function, the control supports milling tools, drilling tools and turning tools. **Further information:** "Measuring the tool by scratching", Page 419
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:** "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258
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 411

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 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 396 **Further information:** "Preset table *.pr", Page 520
ⓘ	Display help graphics for the selected touch probe function
Y+	Select the probing direction
	actual position capture
	Position Manually approach and probe positions on a straight surface You position the workpiece touch probe and start the probing process manually.
A	**Circle center A** Automatically approach and probe positions on a stud or in a hole You manually pre-position the touch probe to the first probing point and define the parameters for the selected touch probe function. When you start the touch probe function, the control automatically positions and performs probing. After the defined number of touch points has been reached, the control determines the center of the stud or the hole. After the last probing 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.
M	**Circle center M** Automatically approach and probe positions on a stud or in a hole You position the workpiece touch probe and contact the individual probing points manually. If you select the **Assume measurement result** button, the control determines the center of the stud or hole.
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 203

Icon or button	Meaning
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic handwheel", Page 535
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 432

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 404

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 405

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 242

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 during 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 242
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 control resets the last touch point. Only with the **Circle center M** measuring method
Assume measurement result	The control determines the circle center of the probed stud or the probed hole. The control enters the circle center in the **Measuring** column. Only with the **Circle center M** measuring method after at least three touch points.
Compensate the active preset	The control sets the active workpiece preset to the entered nominal value. If the workpiece preset is locked, the control shows a window with a confirmation request. **Further information:** "Preset table *.pr", Page 520
Correct the datum	The control opens the **Correct the datum** window. In the **Nominal value** area, enter the desired distance of the measuring result from the workpiece datum. Based on this information, the control calculates the required table value relative to the active workpiece preset. The control enters the calculated value in the selected line of the datum table. **Further information:** Programming and Testing User's Manual
Align rotary table	The control moves the rotary axes to compensate for the measured misalignment.
Correct the pallet reference point	The control sets the active pallet datum to the entered nominal value. If the pallet preset is locked, the control shows a window with a confirmation request. **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 233

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.

If you accept the changes and the active preset is locked, the control shows a window with a confirmation request.

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 functions

After executing any touch-probe function, 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 probe functions in a row, the control stores the measured values below each other.

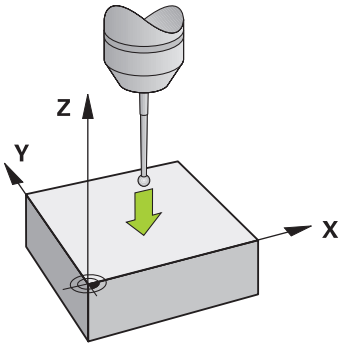
17.1.1 Setting a preset in a linear axis

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 **Position (POS)**
- > 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 sets the active preset to the entered nominal value.

> Using the **Position (POS)** touch probe function, you can probe in up to three axes.

- Select **Exit probing**
- > The control closes the **Position (POS)** touch probe function.



17.1.2 Setting the circle center point of a stud by means of 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)** touch probe function.

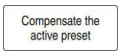
- If necessary, check the values of the active preset or choose a different preset
- Position the workpiece touch probe at the desired probing position in the workspace

- Select **Type of contour** (e.g., stud)
- Enter **Diameter** (e.g., 60 mm)
- Enter the **Safety clearance (min. value = SET_UP)** if required

> The control suggests the sum of the value in the **SET_UP** column of the touch probe table and the ball tip radius as a safety clearance.

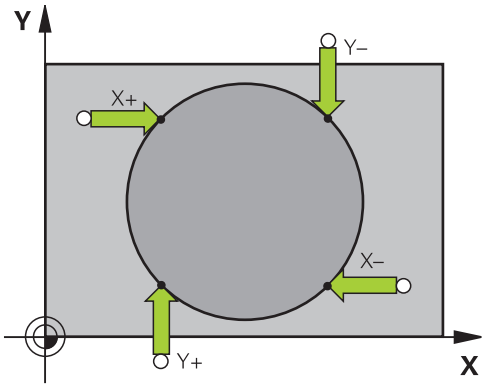
- Enter **Starting angle** (e.g., –180°)
- Enter **Angular length** (e.g., 360°)
- Enter **No. of touch points** (e.g., 4)
- Position the workpiece touch probe at the desired probing position next to the workpiece and below the workpiece surface
- If necessary, turn the feed-rate potentiometer to zero

- 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 probed axes (e.g., **0**)



- Select **Compensate the active preset**
- The control sets the active preset to the entered nominal value.

- Select **Exit probing**
- The control closes the **Circle (CC)** touch probe function.


Y
Y–
X+
X–
Y+
X

17.1.3 Determining and compensating for the basic rotation of a workpiece

To probe the basic rotation of a workpiece:

- Select the **Manual** operating mode

- Call the workpiece touch probe as a tool

- Select the **Setup** application

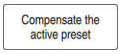
- Select **Rotation (ROT)**
- The control opens the **Rotation (ROT)** touch probe function.

- If necessary, check the values of the active preset or choose a different preset
- Position the workpiece 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 operation and limits the subsequently selectable probing directions.
- Position the workpiece touch probe at the second probing position in the workspace

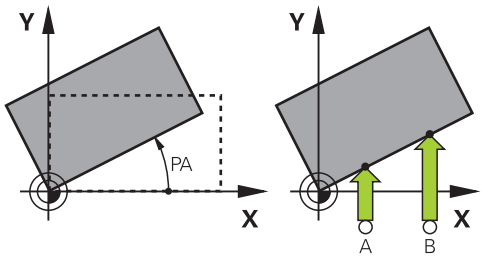
- Press the **NC Start** key
- The control executes the probing operation and then shows the measurement results.
- Activate the **Basic rotation** toggle switch as needed



- Select **Compensate the active preset**
- The control transfers the determined basic rotation with respect to the tool axis to, for example, the **SPC** column of the active row of the preset table.

- The control displays an icon in the **Positions** workspace to show that a basic rotation is defined.

- Select **Exit probing**
- The control closes the **Rotation (ROT)** touch probe 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** of the preset table.

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 also use mechanical probes or tool touch-off for all manual touch probe functions with manual probing methods.

For this, the control provides the **Apply position** button.

To determine a basic rotation with a mechanical probe:

- Select the **Manual** operating mode
- Insert the tool (e.g., analog 3D probe or feeler lever gage)

- Select the **Setup** application

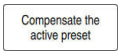
- Select **Rotation (ROT)**

- 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 **Apply position**
- The control saves the current position.
- Move the mechanical probe to the next position to be captured by the control.

- Select **Apply 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, for example, the **SPC** column of the active row of the preset table.

- Select **Exit probing**
- The control closes the **Rotation (ROT)** touch probe 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** of the preset table.

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.
- In the touch probe functions, you can switch between the measuring methods **Position**, **Circle center A** and **Circle center M** as desired. With the **Intersection point (P)** function, you can switch the measuring method after two completed measurements.
- 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 291
- 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 194
- 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 workpiece 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
  - Changes to the **TRACK** parameter
- If the touch point is not reached during the probing process, the control will display a warning. The probing process can be continued with **NC Start**.

Definition

Spindle tracking
If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

17.2 Calibrating the workpiece touch probe

Application

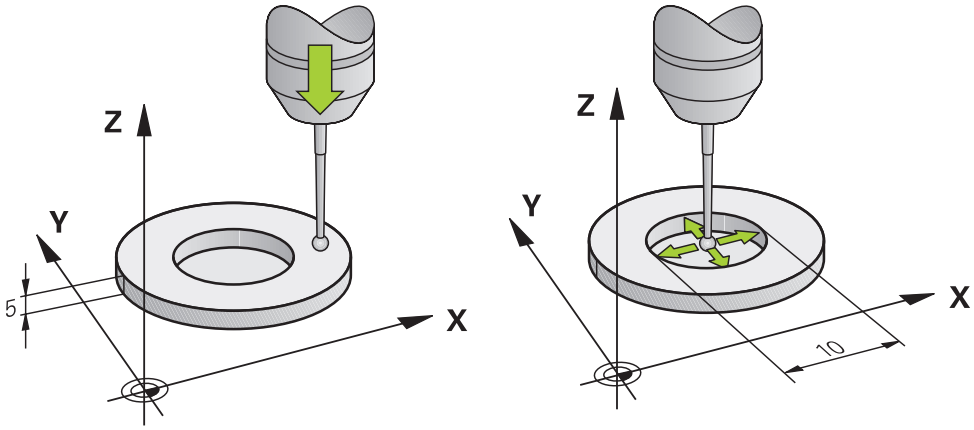
The touch probe must be calibrated in order to enable determining precisely the actual switching point of a 3D touch probe. Otherwise, the control cannot ascertain exact measuring results.

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 506
- 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 calibration ring 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 185

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.

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 positive 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 506

When calibrating the length, the control aligns the touch probe with the calibration angle defined in the **CAL_ANG** column.

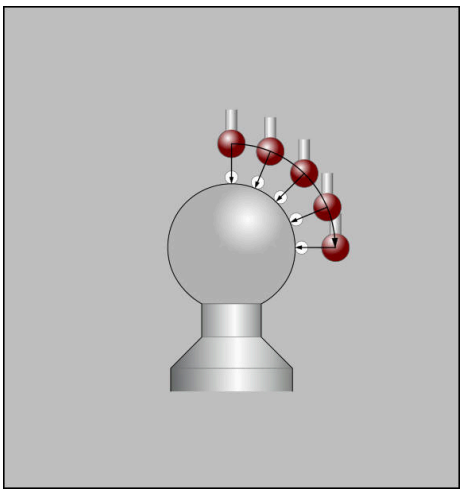
While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

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 234
- 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**)
- Insert the workpiece touch probe

- Select the **Setup** application

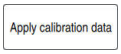
- Select **Calibrate touch probe**

- If necessary, check the values of the active preset or choose a different preset

- Select measuring method **Length**:
- > The control displays the current calibration data.
- Enter a **Position of reference surface** (e.g., **0**)
- Position the workpiece touch probe close to the surface of the overmilled area

> **i** 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 operation 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** touch probe function.

17.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a calibration ring in the radius:

- Clamp the calibration ring on the machine table (e.g., with clamps)

- Select the **Manual** operating mode
- Position the workpiece touch probe in the hole of the calibration ring

> Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe using the largest circumference of the calibration sphere.

- Select the **Setup** application

- Select **Calibrate touch probe**

- If necessary, check the values of the active preset or choose a different preset

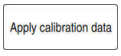
- Select measuring method **Radius**:

- Select calibration standard **Calibration ring**
- Enter the **Diameter** of the calibration ring
- Enter the **Safety clearance (min. value = SET_UP)** if required
- Enter **Starting angle**
- Enter **No. of touch points**

- Press the **NC Start** key

> The workpiece 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 touch probe in the tool table.

- Select **Exit probing**

> The control closes the **Calibrate touch probe** 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 calibration sphere 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

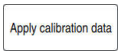
- Select measuring method **Radius**:

- Select calibration standard **Calibration sphere**
- Enter the **Diameter** of the sphere
- Enter the **Safety clearance (min. value = SET_UP)** if required
- Enter **Starting angle**
- Enter **No. of touch points**

- Press the **NC Start** key

> The workpiece 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**

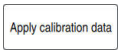
> The control stores the calibrated radius of the touch probe in the tool table.

> The control shows the **3D calibration** measurement method.

- Select measuring method **3D calibration**:
- Enter **No. of touch points**

- Press the **NC Start** key

> The 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** 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 506

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 387
- 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: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 258

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 507
- 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 404
- 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:

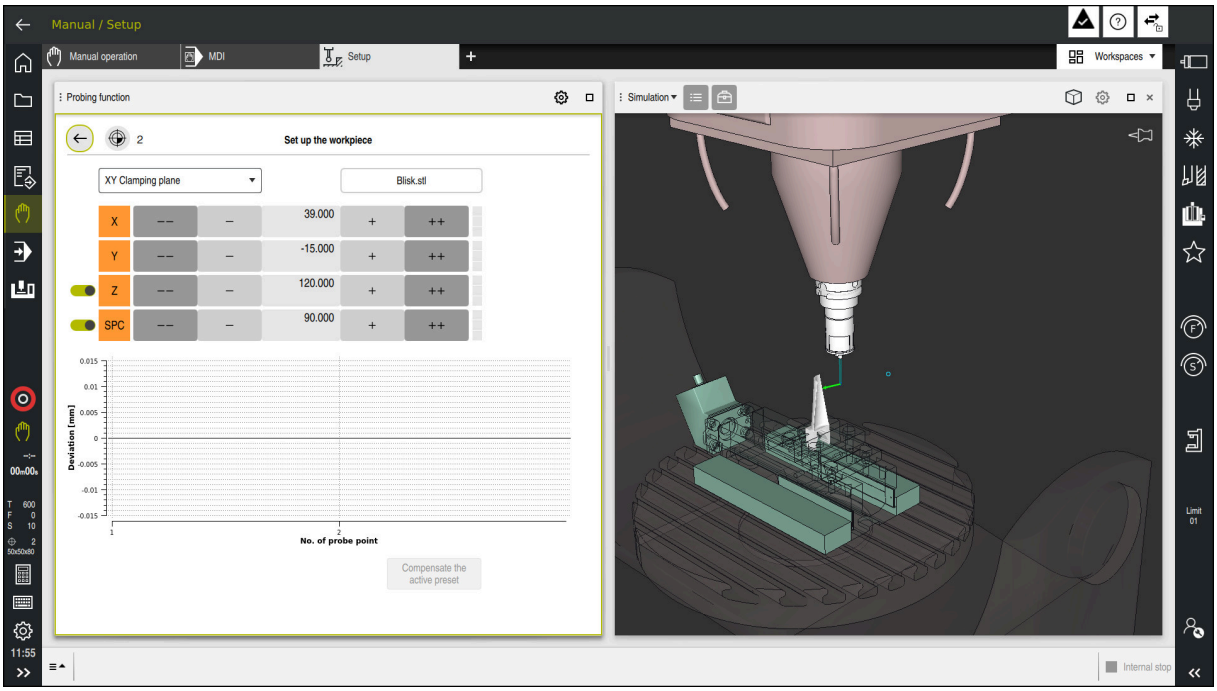
  You cannot tilt before setting up. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 238

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.

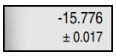
    > Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    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 415
	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.

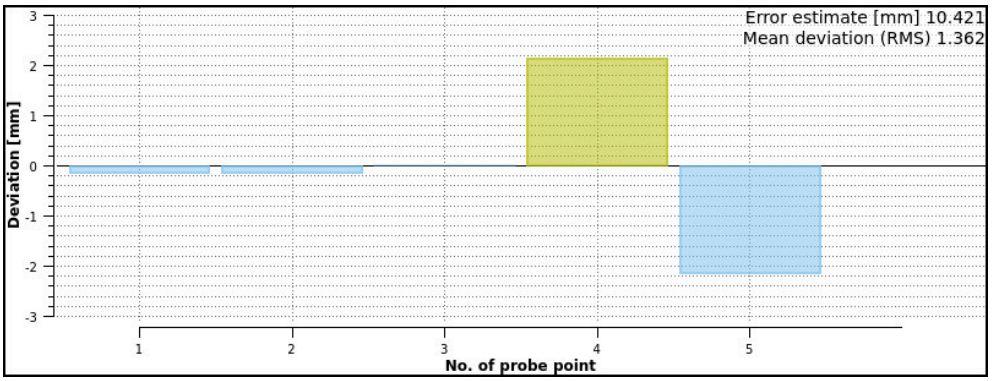
The error estimate diagram shows for each probe point the distance of the 3D model from the real workpiece.

Further information: "Error estimate diagram", Page 416

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:

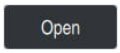
- Affix a real workpiece in the machine's working space

- 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-**
- ▶ Select **Compensate the active preset**

Compensate the active preset

- > The control saves the determined values in the active row of the preset table.
- ▶ Exit the **Set up the workpiece** function

←

Notes

NOTICE

Danger of collision!

To probe the clamping situation in the machine exactly, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision.

- ▶ Calibrate the workpiece touch probe at regular intervals
- ▶ Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the workpiece.
- Collisions might be more easily detected, if a tool carrier is assigned to the workpiece touch probe.
- HEIDENHAIN recommends probing check points for one axis direction on both sides of the workpiece. As a result, the control will correct the position of the 3D model in the simulation uniformly.
- 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 387
- 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 478

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

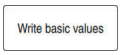
- Select the **Measure tool** touch probe function

- 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** touch probe 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!

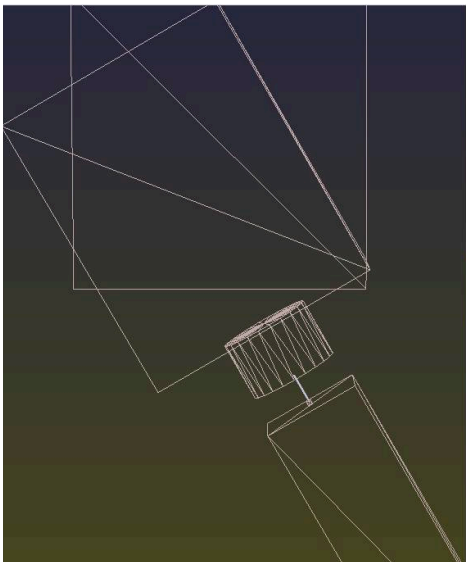
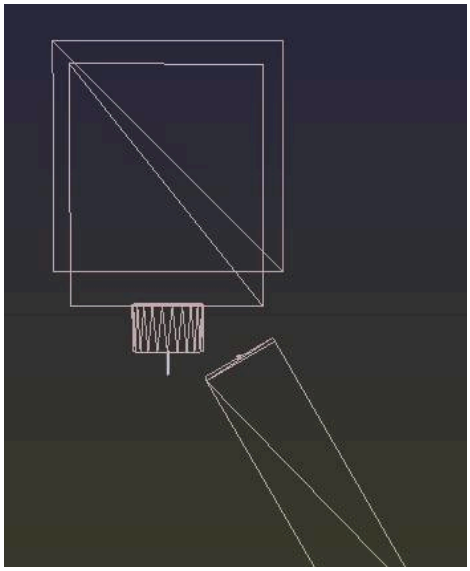
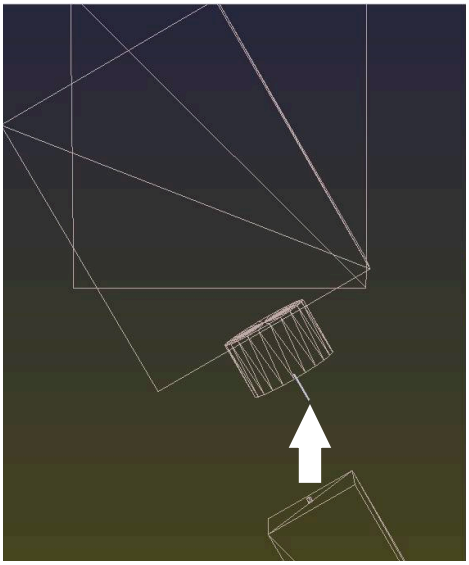
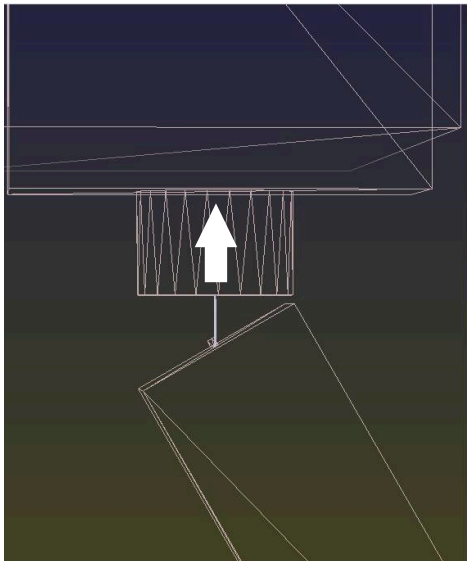
While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

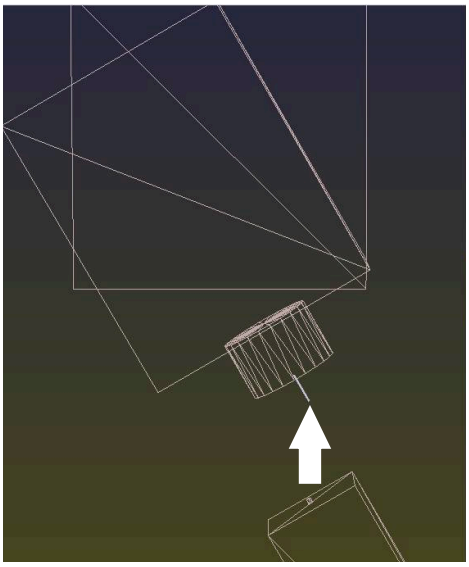
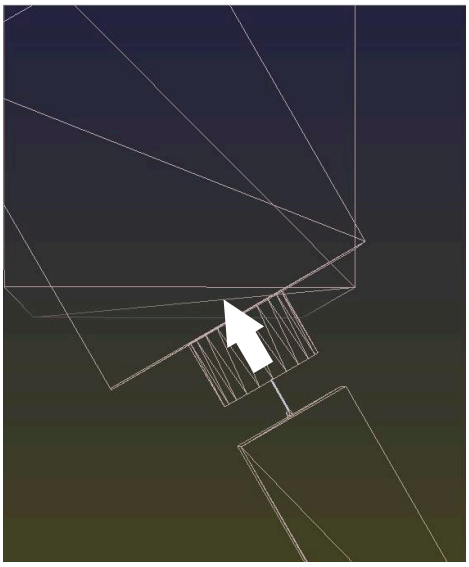
- Carefully move the axes in the **Manual** operating mode

If the touch probe sends a stable signal within these 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 383
- 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 438

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 of the TNC bar", Page 133

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 291
- **Document**
  Further information: Programming and Testing User's Manual
- **Positions**
  Further information: "The Positions workspace", Page 127
- **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 135
- **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 450
F LIMIT	Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 431

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 556 ■ **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 556 ■ **Feed F LIMIT** Use this function to activate a feed-rate limit and define its value. **Further information:** "Feed rate limit F LIMIT", Page 431 ■ **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 284
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 287

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
Automatic program start	Starts machining at a defined time automatically **Further information:** "Automatic program start", Page 447
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 290
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 437
Handwheel	If **Manual traverse** is active, you can activate and deactivate the handwheel with this switch. The keys of the machine operating panel cannot be used for traversing while the handwheel is active. **Further information:** "Manual traverse during an interruption", Page 437
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 437
Approach position	Return to contour after manual traverse of the machine axes during an interruption **Further information:** "Returning to the contour", Page 446
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 438
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. **Further information:** "The Retract application", Page 452
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

Icon or button	Meaning
Tools	The control opens the **Tool management** application in the **Tables** operating mode. **Further information:** "Tool management ", Page 203
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.
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 the **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 125

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 **M1**)
- 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 379

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 of the TNC bar", Page 133

Below are some of the functions you can execute in an interrupted state:

- Select the **Manual** operating mode
- The **Manual traverse** function
- 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 **/**

> You can use the **Manual traverse** function also when program run is stopped.

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 program run in the NC block containing one of the following inputs:

- Programmed stop **STOP**
- 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 program run with an **Internal stop**, you must start program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of 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 438

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 446

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 programs 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 438
- 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 program run, such as handwheel superimpositioning (#44 / #1-06-1).

  Further information: "Global Program Settings (GPS) (#44 / #1-06-1)", Page 291
- 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 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 432

Requirement

- The **Program** and **Simulation** workspaces are both opened
  In the **Editor** operating mode you need both workspaces to use the function.

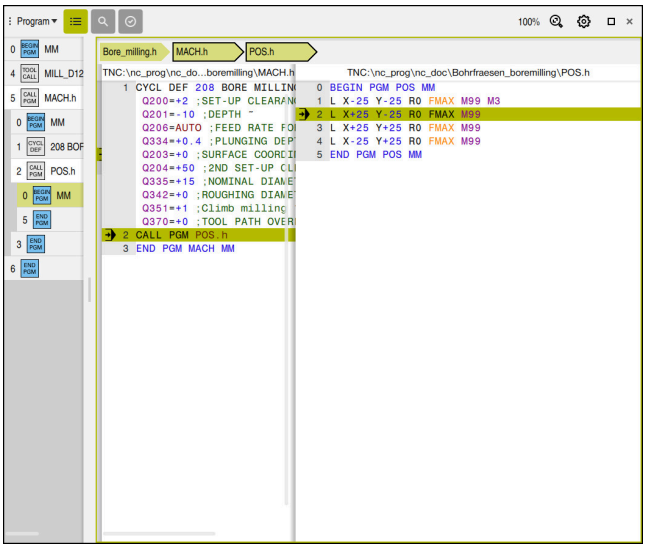
Description of function

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

You can interrupt program run and move the machine axes manually, for example to reduce the risk of injury during workpiece measurement.

Related topics

- Manual traverse of machine axes
  Further information: "Moving the machine axes", Page 172
- Tilting the working plane manually (#8 / #1-01-1)
  Further information: Programming and Testing User's Manual

Description of function

When you select the **Manual traverse** button, you can move the axes of the machine.

Further information: "Using axis keys to move the axes ", Page 172

If **Manual traverse** is active, the operating mode's icon in the control bar changes.

Tilt the working plane window (3D ROT) (#8 / #1-01-1)

Use the **Tilt the working plane (3D ROT)** window to assign the reference system for traversing the axes.

In the **Tilt the working plane (3D ROT)** window, you can select the following functions:

Symbol	Function	Meaning
	**M-CS machine**	Traversing in the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 218
	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 223
	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 226
	**T-CS tool**	Traversing in the tool coordinate system **T-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 226

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.

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

- Refer to your machine manual.

  On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.
- Using the optional machine parameter **userControl** (no. 144101), the machine manufacturer defines if the control will display the **Handwheel** toggle switch or not.

  If the **Handwheel** toggle switch is available, the control displays an additional area in its machine settings. You can define whether the machine operating panel or the handwheel is active by default.

  Further information: "The Machine Settings menu item", Page 581

  If this machine parameter has been defined with **FALSE** or not at all, the machine manufacturer determines the user control that is active by default.
- While the **Handwheel** toggle switch is active, you cannot move the axes with the machine operating panel.
- Refer to your machine manual.

  Depending on the machine, the control indicates that a handwheel is active (for example, with a handwheel symbol next to the selected axis in the **Positions** workspace or via a message in the information bar).

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

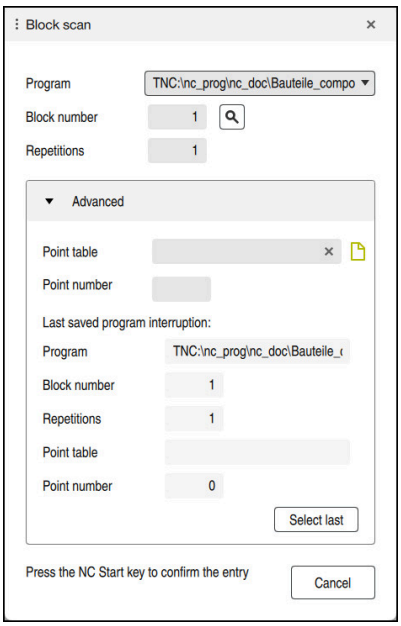
If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

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 441
- Multi-level block scan in subprograms and touch probe cycles
  Further information: "Performing a multi-level block scan", Page 442
- Block scan in a point table
  Further information: "Block scan in point tables", Page 443
- Block scan in pallet programs
  Further information: "Block scan in pallet tables", Page 444

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** if required
- Enter **Block number**
- Enter the **Repetitions** if required

- 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 until 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 your own selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 447

- 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 441

- 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 where you wish to start

- Press the **NC Start** key
- The control starts the block scan and calculates until 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 your own selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 447

- 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

- **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 until 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 your own selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 447

If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired start-up point in the **Point number** field. The first point in the point pattern has the number 0.

Further information: User's Manual for Machining Cycles

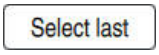
Block scan in pallet tables

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** if required
- Enter **Block number**
- Enter the **Repetitions** if required

- 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 until 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 your own selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 447

> If 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-level block scan 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 at multiple levels

- The export version of the control, TNC7 E, can move at most up to four axes at the same time. If more than four axes need to be moved as part of an NC block, the control will issue an error message. When the control reads such an NC block during the **Block scan**, it will also display an error message.
- The control only displays the dialogs required by the process in the pop-up window.
- 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 143
- 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 437
- The **Block scan** function

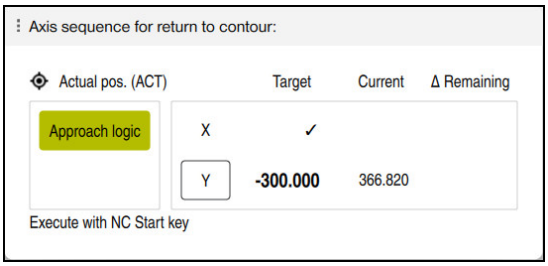
  Further information: "Block scan for mid-program startup", Page 438

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 447

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 447

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

Approach position

- ▶ 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 check mark for **Target**.
- ▶ Position the remaining axes
- > When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

Approach position

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the manual axis (e.g., **W**)
- ▶ Position the manual axis to the value shown in the window
- > When a manual axis with encoder has reached the position, the control automatically clears the value.
- ▶ Select **Axis in position**
- > The control saves the position.

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

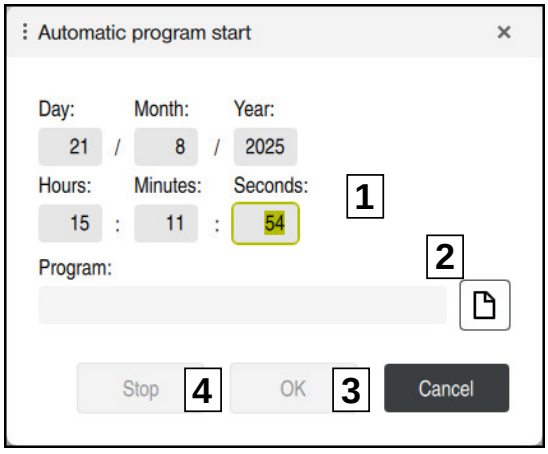
- The function must be enabled by your machine manufacturer.
  In the machine parameter **autoStartEnabled** (no. 100701), the machine manufacturer defines whether an automatic program start is possible.
- Machine is switched on
- No NC program is currently running on the machine
  If a program is already 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.

When you select the **Automatic program start** button, the control displays 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 105

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 of the TNC bar", Page 133

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 453
- **Positions**

  Further information: "The Positions workspace", Page 127
- **Status**

  Further information: "The Status workspace", Page 135

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 431

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 216
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.
- > In the **Start** workspace, the control provides the **Power interrupted** column.

- ▶ Activate the **Retract** toggle switch

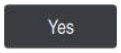
- ▶ Select **OK**
- > The control compiles the PLC program.

- ▶ 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 462

Further information: "The Form workspace for tables", Page 470

You can select various functions by using the context menu (e.g., **Copy**).

Further information: Programming and Testing User's Manual

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. The machine manufacturer can also change the colors in the control's GUI.

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 selects the line number that you 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 200 ■ **Line number (Tool number?)** ■ **Number of rows** ■ **Index** **Further information:** "Indexed tool", Page 194 ■ **Append** Append rows at the end of the table **Further information:** "Tool management ", Page 203
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 203
Import	The control imports table contents from a CSV file (e.g., tool data). **Further information:** "Importing and exporting of table contents", Page 473
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 520
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 523

Refer to your machine manual.
If necessary, the machine manufacturer adapts the buttons.

If at least one table row is marked, the control contains the following buttons in the action bar:

- **Cut**
- **Copy**
- **Reset**

  Only for tables that also provide the **Reset row** button
- **Delete**
- **Select all**
- **Cancel marking**
- **Export**

  Export data of the selected lines as a CSV file

  Further information: "Importing and exporting of table contents", Page 473
- **Cancel**

19.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Activate **Edit**
- > The control enables the values for editing.

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 have, for example, processed tables in a text editor, the control provides 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

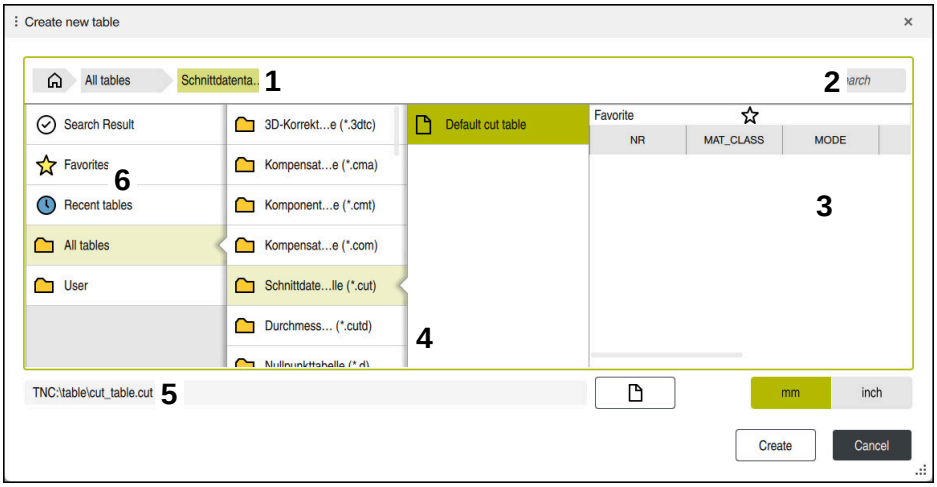
Application

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 change the path or use the history to open a previous path.

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

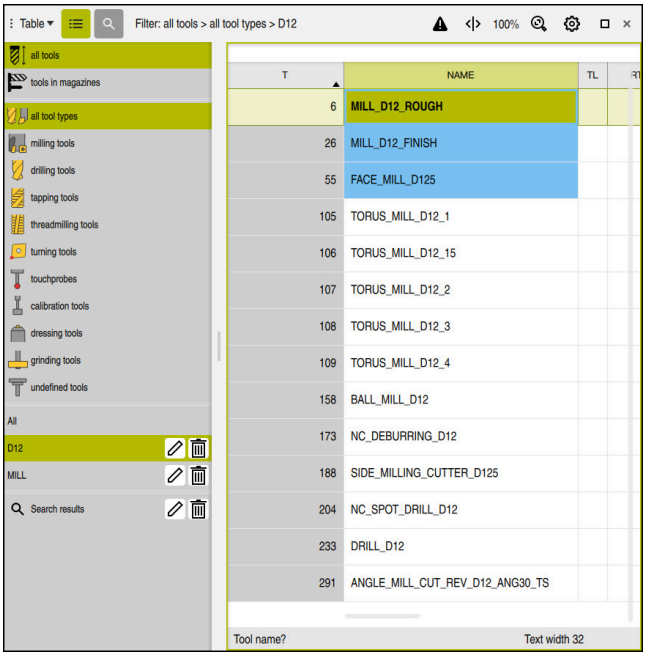
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer 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.

If you select a cell, the control displays the following information on the selected parameter in the dialog bar:

- Name
- Text width or input range
- Unit of measure as needed

Refer to your machine manual.

If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Icons and shortcuts

The **Table** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
	Open or close the **Filter** column **Further information:** "The Filter column in the Table workspace", Page 464
 **[CTRL] + [F]**	Open or close the **Search** column **Further information:** "The Search column in the Table workspace", Page 466
	■ Activate or deactivate **"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 467
	Open the **Search** column and edit the selected filter Only in the **Filter** column **Further information:** "The Search column in the Table workspace", Page 466
	Delete the selected filter Only in the **Filter** column **Further information:** "The Filter column in the Table workspace", Page 464
[CTRL] + [A]	Mark all rows
[CTRL] + [SPACE]	Mark the active row or end the marking function
[SHIFT] + [UP]	Additionally mark the row above
[SHIFT] + [DOWN]	Additionally mark the row below

The Filter column in the Table workspace

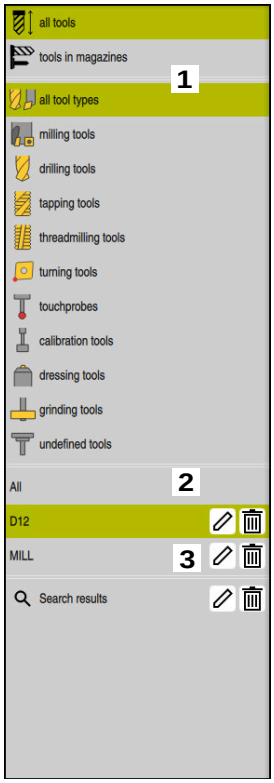
The control provides default filters for the following tables:

- **Tool management**
- **Pocket table**
- **Presets**
- **Tool table**

For all other tables, you can create user-defined filters.

Further information: "User-defined filters", Page 466

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 466
3 **Search results**
   Results of the **Search** column
   Further information: "The Search column in the Table workspace", Page 466

When you tap or click a filter once, the control activates only the selected filter in the respective group.

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 466

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. The machine manufacturer can also change the colors in the control's GUI.

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**
- **Calibration tools**
- **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 466

The control only shows this filter group when you create a user-defined filter. In addition to the user-defined filters, the control displays the **All** filter option.

Further information: "The Filter column in the Table workspace", Page 464

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 466

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: "The Filter column in the Table workspace", Page 464

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. The machine manufacturer can also change the colors in the control's GUI.

Settings in the Table workspace

In the **Tables** window, you can influence the contents shown in the **Table** workspace.

The **Tables** window consists of the following areas:

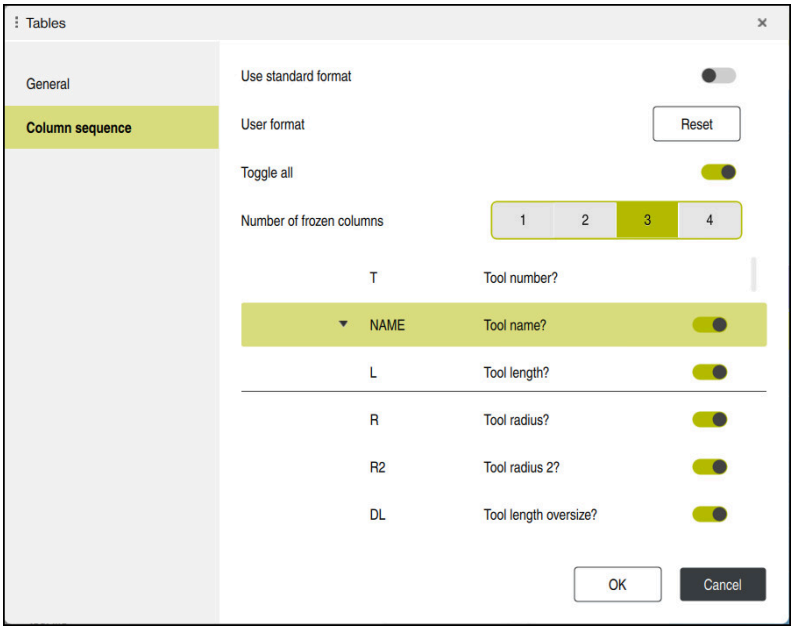
- **General**
- **Column sequence**

The General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** toggle switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control moves the cursor synchronously in the **Form** workspace.

The Column sequence area



The **Tables** window

The **Column sequence** area contains the following settings:

Setting	Meaning
Use standard format	If you activate the toggle switch, the control shows all table columns, indicating them in the standard sequence. If you deactivate the toggle switch, the control restores the previous setting.
User format	If you select the **Reset** button, the control resets the adaptations to the settings of the standard format.
Toggle all	If you activate the toggle switch, the control shows all table columns. If you deactivate the toggle switch, the control hides all table columns. The first column in each table cannot be hidden.
Number of frozen columns	You define how many table columns the control freezes at the left table edge. You can freeze up to four table columns. These table columns will remain visible even when you navigate further to the right within the table.
Columns of the currently opened table	The control displays all table columns below each other. Use the toggle switches to separately hide or show each table column. The control displays a line below the selected number of frozen columns. When you select a table column, the control displays up and down arrows. Use these arrows to change the sequence of the columns. The respective first column in the table cannot be shifted.

The settings in the **Column sequence** area only apply to the currently opened table.

Virtual columns

In the **Table** workspace, the control can display virtual columns. Virtual columns are not included in the table files but contain calculated values from other data.

The control contains the following virtual columns:

Column	Meaning	Application
MAGAZIN	The control shows whether the tool is currently in the magazine or in the spindle.	■ Tool management ■ Pocket table
TOOL_LIFE	Information on the tool life: ■ **+**: The current tool life **CUR_TIME** is at least 5 min below the maximum tool life **TIME2**. ■ **-**: The current tool life **CUR_TIME** is less than 5 min below the maximum tool life **TIME2**. ■ **X**: The tool has reached the maximum tool life **TIME2**. ■ **?**: No value defined for the maximum tool life **TIME2**	■ Tool management ■ Pocket table

Refer to your machine manual.

The machine manufacturer can modify virtual columns and define further columns.

Notes

- If you insert the contents of the clipboard while editing is inactive, the control displays the **Editing disabled. Enable?** prompt. If you select **Yes**, the control inserts the copied contents.
- Refer to your machine manual.

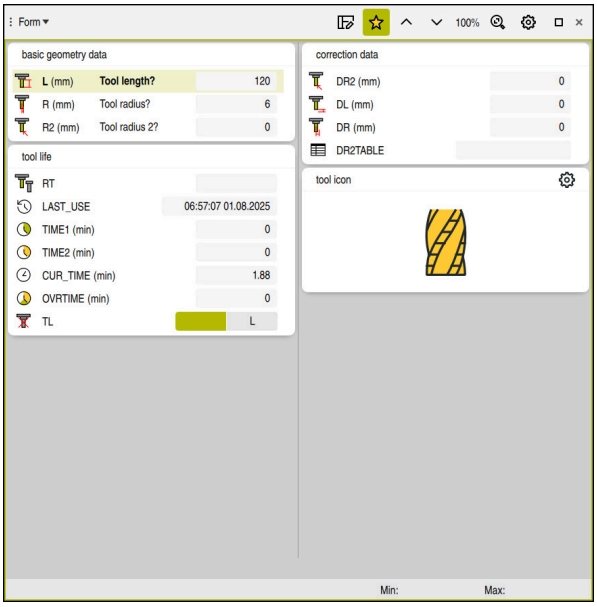
  Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.
- Since the values in virtual columns are not included in the table file, you cannot read these values with functions such as:
  - NC functions (e.g., **TABDATA READ**)
  - **HEIDENHAIN DNC** (#18 / #3-03-1)
  - **OPC UA NC Server** (#56-61 / #3-02-1*)

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:

- An icon, if applicable
- Name
- Text width or input range
  Only in the dialog bar
- Unit of measure as needed
  Only in the dialog bar
- 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).

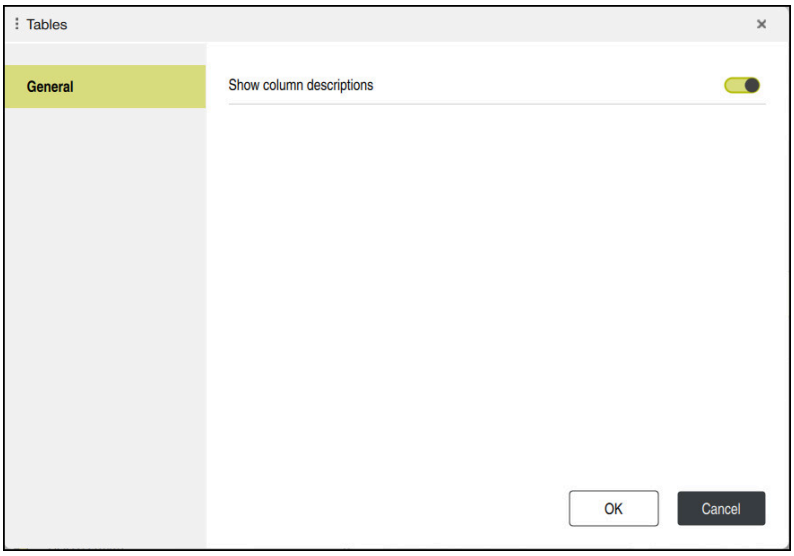
Icons

The **Form** workspace contains the following icons or shortcuts:

Icon or shortcut	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
	**Favorites** In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites. If the icon is not active, the control shows all areas.
[SHIFT] + [UP]  **[SHIFT] + [DOWN]**	**Navigate** Navigate between table rows
100%	Current size of the content Open or close the **Scale** selection menu
	**Reset scaling** Set font size of the content to 100%
	**Settings** ■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 472 ■ 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 472 ■ 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

- In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
- 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 200
- 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
- Refer to your machine manual.

  Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.

19.5 Importing and exporting of table contents

Application

You can import the contents of specific tables into the control and export them from the control. This avoids manual editing efforts and possible typing errors.

Importing tool data is particularly useful in connection with a tool presetter. Via exported contents you can, for example, transfer the tools and presets of the control to the CAM system.

The control uses CSV files for exporting and importing of table contents.

Related topics

- Overview of file types
  Further information: Programming and Testing User's Manual
- Fundamentals of the **Tool management** application
  Further information: "Tool management ", Page 203

Description of function

Importing of data is possible with all tables that contain the **Import** button in the function bar.

To export contents, you have to mark all desired table rows. If a table does not permit exporting, the **Export** button is grayed out in the action bar.

The CSV file has the following structure:

- The first line contains the column names of the table being exported.
- The other lines contain the exported contents. The contents are arranged in the same sequence as the first line. Decimal numbers are separated by a point.

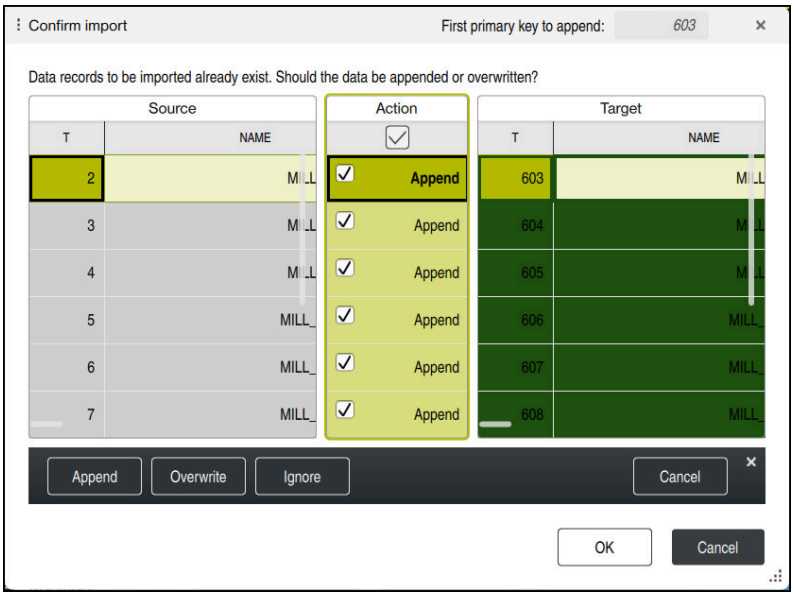
For compatibility with earlier controls, the control provides the following formatting options for exporting:

- **TNC7 (semicolon-separated)** writes the values between double quotation marks and separates the values by semicolons.
  Most table calculation programs use this formatting.
- **iTNC 530 / TNC 640 (comma-separated)** writes the values partly between brackets and separates the values by commas

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

The Confirm import window

When you select a CSV file for importing, the control will display the **Confirm import** window.



The **Confirm import** window with table rows to be appended

The window contains the following areas:

Area	Meaning
Source	The control shows the values of the CSV file that are to be imported.
Action	If you double-tap or double-click a row, the following actions for this row can be selected: ■ **Append** Append row at end of table ■ **Overwrite** Import row with line number of the CSV file ■ **Ignore** Do not import row of the CSV file Clicking the checkbox in the header selects all table rows. You can select the desired action for all rows that have been marked in the action bar.
Target	The control shows the table values that will result from the currently selected action after importing. Depending on the selected action, the control shows the cells in the following colors:

Color	Meaning	Action
Yellow	The content of the source differs from the content of the target table.	**Ignore**
Dark gray	The table will not contain a row with this number after importing.	**Ignore**
Dark green	The rows will be added in the target file.	**Append**
Red	The current table content will be overwritten after importing.	**Overwrite**

The control displays an input field in the title bar of the window. If **Append** is selected, you can optionally enter a row number that will be the start number of the imported table rows. This allows you to define new tool numbers, for example.

19.5.1 Exporting table contents

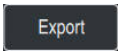
To export table contents:

- Select the **Tables** operating mode
- Select the table (e.g., **Tool management**)
- Select the row to be exported

- Select **Mark row**

> The control marks the selected line and opens the action bar.

- Select further rows, if required



- Select **Export**

> The control opens the **Save as** window.

- Select a path
- Enter the file name
- Select the formatting

- Select **Save**

> The control saves the CSV file using the defined path.

If you export tools from the **Tooling list** (#93 / #2-03-1) table, you have to use the context menu instead of the **Mark row** button. The exported CSV file contains all contents of the tool, and not just the columns of the tooling list. This allows you to import this CSV file to the tool management.

Further information: "Tooling list (#93 / #2-03-1)", Page 518

19.5.2 Importing table contents

To import table contents:

- Select the **Tables** operating mode
- Select the table (e.g., **Tool management**)

- Activate **Edit**

> The control enables the table for editing.

- Select **Import**

> The control opens a selection window.

- Select the desired CSV file

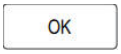
- Select **Import**

> The control opens the **Confirm import** window.

- Click the checkbox in the header of the **Action** area

> The control marks all rows to be imported and opens the action bar.

- Select an action:
  - **Append**: the control adds the contents as new rows at the end of the table.
  - **Overwrite**: the control overwrites the initial contents with the contents from the CSV file.
  - **Ignore**: the control does not import the contents.



- Select **OK**

> The control imports the contents with the selected action.

If you double-tap or double-click each cell in the **Action** area, the following actions can be selected separately for each row.

NOTICE

Caution: Data may be lost!

When overwriting existing table values with the **Overwrite** function, the control will permanently delete the initial values!

- Use this function only with table files that are no longer needed

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 CSV file contains information on the table that the export comes from. Only tables of the same file type allow importing.
- By default, the control saves the export under **TNC:\system\tooltab**.
- To be able to import the CSV file, the column with the line numbers must be available.
- In the **Presets** application, the **Confirm import** window does not include an input field in the title bar. Within the preset table, you can append rows only using the next line number.
- If rows cannot be overwritten when importing CSV files, the control will display an information symbol. If you select the information symbol, the control displays a pop-up window showing the reason for the problem. The control provides a button with the information symbol. It allows you to filter the contents to find all rows with this problem.

19.6 Tool tables

19.6.1 Overview

This chapter contains information about the tool tables of the control.

- Tool table **tool.t**

  Further information: "Tool table tool.t", Page 478
- Turning tool table **toolturn.trn** (#50 / #4-03-1)

  Further information: "Turning tool table toolturn.trn (#50 / #4-03-1)", Page 488
- Grinding tool table **toolgrind.grd** (#156 / #4-04-1)

  Further information: "Grinding tool table toolgrind.grd (#156 / #4-04-1)", Page 492
- Dressing tool table **tooldress.drs** (#156 / #4-04-1)

  Further information: "Dressing tool table tooldress.drs (#156 / #4-04-1)", Page 503
- Touch probe table **tchprobe.tp**

  Further information: "Touch probe table tchprobe.tp", Page 506

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 203

19.6.2 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 203
- Tool parameters

  Further information: "Tool parameters", Page 191

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

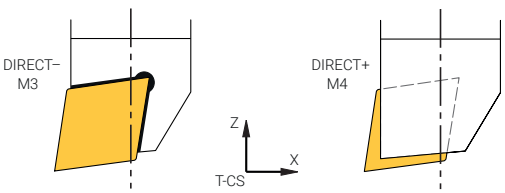
The **Tool_management** application shows the following parameters of the tool table:

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 194 This parameter applies to all tools, regardless of technology.
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 194 This parameter applies to all tools, regardless of technology.
L 	**Tool length?** Length of tool, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 185
R 	**Tool radius?** Tool radius, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 185
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
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**
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**

Parameter	Meaning
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**
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.
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 194 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
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.

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.
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.
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 200 **Further information:** "Tool management ", Page 203 This parameter applies to all tools, regardless of technology. Selection using a selection menu
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 193 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool. **Further information:** "Indexed tool", Page 194 This parameter applies to all tools, regardless of technology.
DOC	**Tool description** This parameter applies to all tools, regardless of technology.
PLC	**PLC status?** Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology.

Parameter	Meaning
LCUTS 	**Tooth length in the tool axis?** Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring.
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).
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.
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.
ANGLE 	**Maximum plunge angle?** Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles.
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)
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
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

Parameter	Meaning
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)
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)
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)
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**). 

Parameter	Meaning
R-OFFS 	**Tool offset: radius?** Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement. **Further information:** 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)
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)
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)
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)
NMAX 	**Maximum speed [rpm]** Limitation of spindle speed for the programmed value including control by the potentiometer. For grinding tools (#156 / #4-04-1), you can also define a maximum permitted cutting speed with the **V_MAX** parameter. The control will use the smaller of the two values as the limit. **Further information:** "Parameters of the grinding tool table toolgrind.grd", Page 495

Parameter	Meaning
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
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 506 Selection by means of a selection window
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
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
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 280 Selection by means of a selection window
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 286
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 288
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 288
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.

Parameter	Meaning
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table **Further information:** "Pocket table tool_p.tch", Page 511 Refer to your machine manual. This parameter applies to all tools, regardless of technology.
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 290
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 205 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 209 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.
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).

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 510
- 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.
- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.

19.6.3 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 203
- Tool parameters
  Further information: "Tool parameters", Page 191
- 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 478

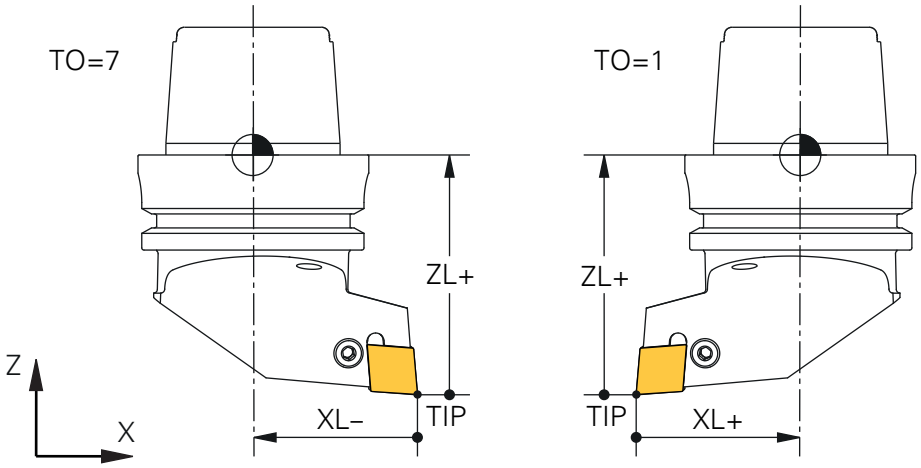
Requirements

- 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 200

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 194 The row number must match the tool number in the **tool.t** tool table.
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 194
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 185
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 185
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 185
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**
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**
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**

Parameter	Meaning
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**.
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**
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**.
ORI 	**Angle of spindle orientation?** Angle of tool spindle for aligning the turning tool
SPB-INSERT 	**Angular offset?** Angular offset for recessing and threading tools with an offset tool holder, spatial angle B Cycle **800 ADJUST XZ SYSTEM** considers the angular offset for the tool angle of inclination and also performs compensation of the B axis. **Further information:** User's Manual for Machining Cycles

Parameter	Meaning
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**.
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 In the **Tool Icon** area of the **Form** workspace, the control shows the angle position corresponding to the tool orientation **TO**. For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **P-ANGLE**.
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.
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
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**
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 201 **Further information:** "Tool management ", Page 203 Selection using a selection menu

Parameter	Meaning
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 226
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 226

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 510
- 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.

19.6.4 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 203
- Tool parameters
  Further information: "Tool parameters", Page 191
- 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 503
- General parameters, regardless of the technology
  Further information: "Tool table tool.t", Page 478

Requirements

- Grinding (#156 / #4-04-1) software option
- Grinding tool is defined in the **TYPE** column of tool management
  Further information: "Tool types", Page 200

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

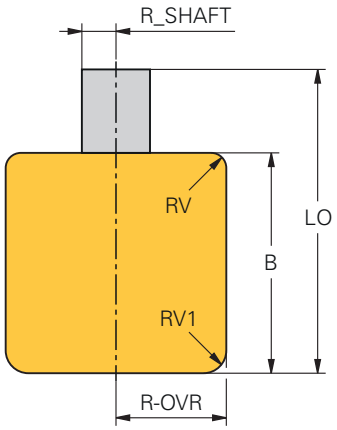
Danger of collision!

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!

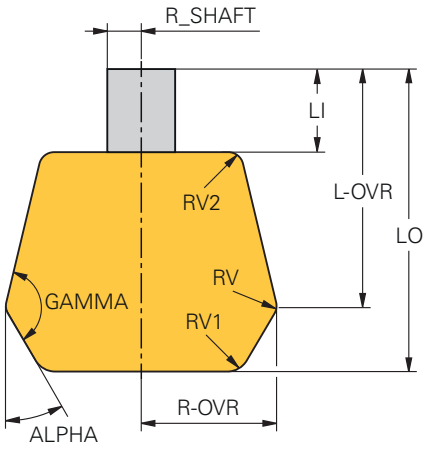
- Edit only freely editable tool management parameters
- Comply with the information about locked parameters in the tool data overview table

The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

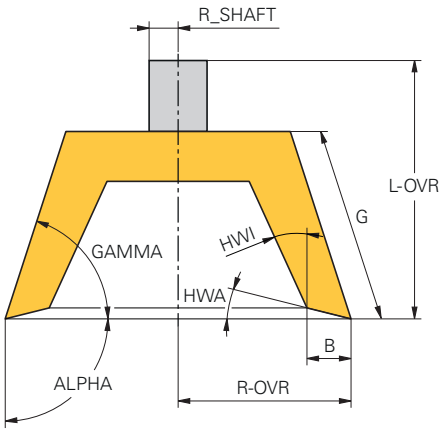
You define the parameters depending on the following grinding tool types:



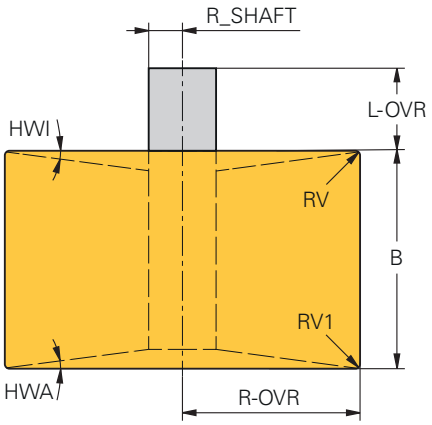
Cylindrical grinding pin



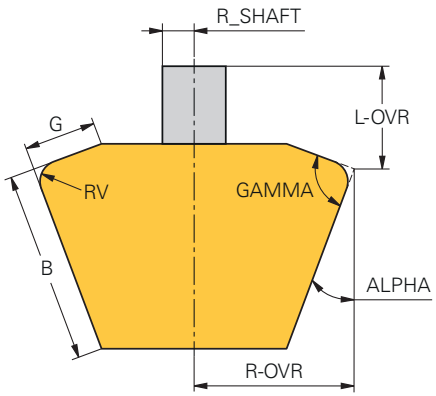
Conical grinding pin



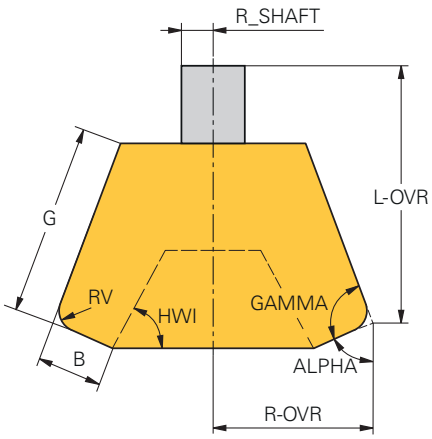
Cup wheel



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 194 The row number must match the tool number in the **tool.t** tool table
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 194
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 202 **Further information:** "Tool management ", Page 203 Selection using a selection menu
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
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
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
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
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

Parameter	Meaning
G 	**Depth** Depth of grinding wheel After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles
ALPHA	**Angle for the slant** After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles
GAMMA	**Angle for the corner** After initial dressing, you can no longer edit this parameter. **Further information:** User's Manual for Machining Cycles
RV 	**Radius at the edge for L-OVR** **Further information:** User's Manual for Machining Cycles
RV1 	**Radius at the edge for LO** **Further information:** User's Manual for Machining Cycles
RV2 	**Radius at the edge for LI** **Further information:** User's Manual for Machining Cycles
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**
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**
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**

Parameter	Meaning
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**
R_SHAFT 	**Radius of the tool shank**
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.
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.
V_MAX 	**Maximum permissible cutting speed** Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. If the **NMAX** parameter is also defined, the control uses the smaller of the two values as the limit.
V	**Current cutting speed** Currently no function
W	**Tilt angle** Currently no function
W_TYPE	**Tilted toward inner or outer edge** Currently no function
KIND	**Type of machining (internal/external grinding)** Currently no function
HW	**Wheel has a relief cut** Currently no function
HWA 	**Angle for relief cut on the outer edge**
HWI 	**Angle for relief cut on the inner edge**

Parameter	Meaning
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
INIT_D_PNR	**Dresser location for initial dressing** Currently no function
INIT_D_DNR	**Dresser number for initial dressing** Currently no function
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**.
STATE	**Setup status** Currently no function
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
A_NR_A	**Dresser number (outer edge dressing)** Currently no function
A_NR_I	**Dresser number (inner edge dressing)** Currently no function
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.
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.
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.
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.

Parameter	Meaning
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.
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.
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
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
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
FORM	**Wheel shape** Selection using a selection menu
A_PL	**Chamfer length at outside**
A_PW	**Chamfer angle at outside**
A_R1	**Corner radius at outside**
A_L	**Length of outside** Currently no function
A_HL	**Length of relief cut, wheel depth at outside**
A_HW	**Angle of relief cut at outside**
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
A_R2	**Angle of departure at outside**
A_G	**Reserve at outside** Currently no function
I_PL	**Chamfer length at inside**
I_PW	**Chamfer angle at inside**
I_R1	**Corner radius at inside**
I_L	**Length of inside** Currently no function
I_HL	**Length of relief cut, wheel depth at inside**
I_HW	**Angle of relief cut at inside**

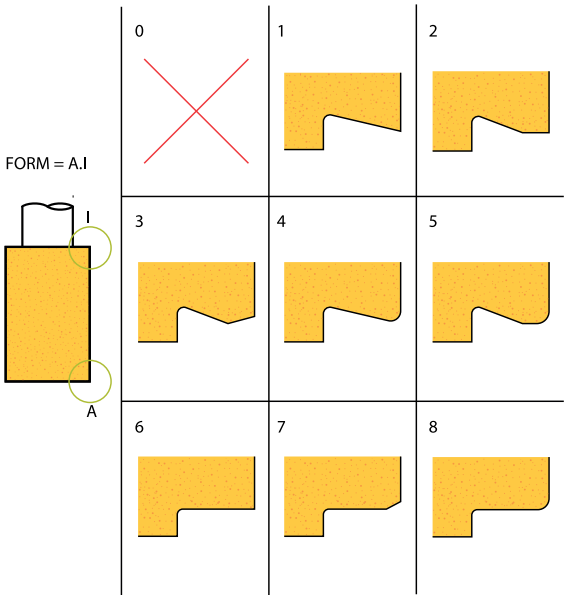
Parameter	Meaning
I_S	**Side depth at inside** Depth of an already existing profile The control automatically corrects the value by the dressed value. Without function if **HWI** is defined
I_R2	**Angle of departure at inside**
I_G	**Reserve at inside** Currently no function
COR_TYPE	**Selection of compensation 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 using a selection menu
COR_ANG	**Inclination angle of dressing tool**

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 115

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 510
- 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 following 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

19.6.5 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 203
- Tool parameters
  Further information: "Tool parameters", Page 191
- 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 492
- General parameters, regardless of the technology
  Further information: "Tool table tool.t", Page 478

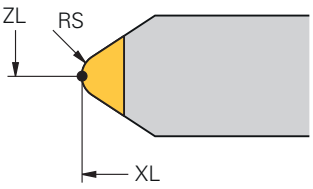
Requirements

- Grinding (#156 / #4-04-1) software option
- Dressing tool is defined in the **TYP** column of tool management
  Further information: "Tool types", Page 200

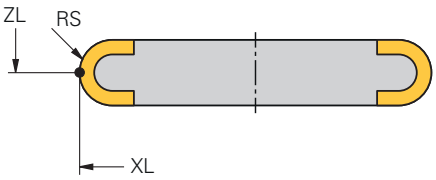
Description of function

The file name of the dressing tool table is **tooldress.drs** and this table must be stored in the folder **TNC:\table**.

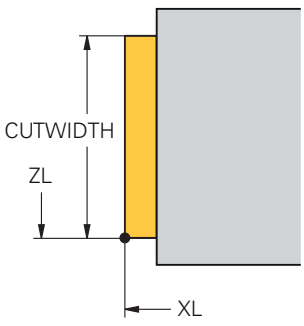
You define the parameters depending on the following dressing tool types:



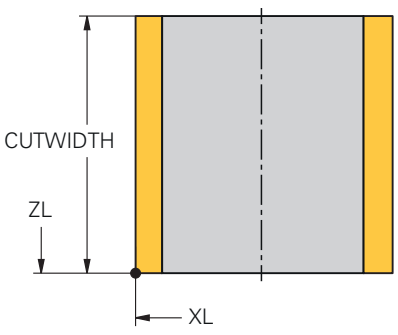
Stationary dresser with radius



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 194 The row number must match the tool number in the **tool.t** tool table.
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 194
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 185
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 185
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 185
DZL 	**Tool length oversize 1** Delta value of tool length 1 for tool compensation Is added to the parameter **ZL**
DXL 	**Tool length oversize 2** Delta value of tool length 2 for tool compensation Is added to the parameter **XL**
DYL 	**Tool length oversize 3** Delta value of tool length 3 for tool compensation Is added to the parameter **YL**
RS 	**Tool tip radius**
DRS 	**Cutter radius oversize** Delta value of the cutter radius for tool compensation Is added to the parameter **RS**

Parameter	Meaning
TO 	**Tool orientation** The control uses the tool orientation to determine the position of the tool's cutting edge.
CUTWIDTH	**Width of tool (plate, roll)** Tool width of the tool types **dressing plate** and **dressing roll**
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 202 **Further information:** "Tool management ", Page 203 Selection using a selection menu
N-DRESS	**Speed of the tool (dressing spindle)** Shaft speed of a dressing spindle or dressing roll

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 510

19.6.6 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 203
- Tool parameters
  Further information: "Tool parameters", Page 191
- Touch probe functions
  Further information: "Touch probe functions in the Manual operating mode", Page 387
- 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** Enter this number in the **TP_NO** parameter of the tool management. The control links the data of the touch probe table to the tool management.
TYPE 	**Selection of the touch probe?** The TS 642 touch probe makes the following values available: ■ **TS642-3**: The touch probe is activated by a conical switch. This mode is not supported. ■ **TS642-6**: The touch probe is activated by an infrared signal. Select this mode. Selection using a selection menu
STYLUS 	**Shape of the stylus** ■ **SIMPLE**: Straight stylus ■ **L-TYPE**: L-shaped stylus If you do not define the parameter, the control uses **SIMPLE** Selection using a selection menu
CAL_OF1 	**TS center misalignmt. ref. axis? [mm]** Depending on the selection of the **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.
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.
CAL_ANG 	**Spindle angle for calibration?** Depending on the selection of the **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**
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.

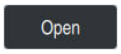
Parameter	Meaning
FMAX 	**Rapid traverse in probing cycle? [mm/min]** Feed rate at which the control pre-positions the touch probe and positions it between the measuring points
DIST 	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will display an error message.
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.
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 Selection using a selection menu
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.
SERIAL 	**Serial number?** The control automatically edits this parameter of touch probes with an EnDat interface.
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**: Interrupting an NC program ■ **EMERGSTOP**: Emergency stop, quick braking of the axes Selection using a selection menu

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 **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.

19.6.7 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.

19.7 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine.
The control needs the pocket table in order to change the tool.

Related topics

- Tool call
  Further information: Programming and Testing User's Manual
- Tool table
  Further information: "Tool table tool.t", Page 478

Requirement

- The tool is defined in the tool management.
  Further information: "Tool management ", Page 203

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
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 478 Selection by means of a selection window
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
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ **S**: Special tool
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

Parameter	Meaning
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
NAME	**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 478
DOC	**Tool comment?** Tool comment from the tool table When you define the tool number, the control will automatically load the tool comment. **Further information:** "Tool table tool.t", Page 478
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.
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual.
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.
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual.
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual.
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual.
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual.
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.

Notes

- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool must be removed from the magazine first.
- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.
- In the **Table** workspace of the pocket table, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.

  Further information: "Virtual columns", Page 469

19.8 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
  Further information: "Tool usage test", Page 212
- 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 478

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 212
- The **Generate tool-usage file** setting is set to **Once** or **Always**
  Further information: "The Machine and Simulation areas", Page 581

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file
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**
TNR	Tool number If the control has not yet inserted a tool, the column contains the value **-1**.
IDX	Tool index
NAME	Tool name
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements)

Parameter	Meaning
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table
BLOCK	NC block number of the tool call
PATH	Path of the NC program, the pallet table, or the tool table
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**.
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**.
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**.
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed
LINENR	Row number of the pallet table in which the NC program is defined

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

19.9 T usage order (#93 / #2-03-1)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created
  Further information: "Creating the tool usage file", Page 212
  Further information: "Tool usage file", Page 514

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 194 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 194 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 511 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool management ", Page 203
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 194
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 514
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 514
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 511

19.10 Tooling list (#93 / #2-03-1)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 212

  Further information: "Tool usage file", Page 514

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 194 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 511 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 205
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 194
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 514
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 511

Note

When you export the contents of the **Tooling list** table, the CSV file contains all contents of the tool, and not just the columns of the Tooling list. This allows you to import this CSV file to the tool management.

Further information: "Importing and exporting of table contents", Page 473

19.11 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 182

Related topics

- Setting and activating presets

  Further information: "Preset management", Page 233

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 221 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**.

Button	Function
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
DOC	Comment
X	X coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 221
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 221
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 221
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 221 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 235
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 221 Has the effect of a 3D basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 235
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 221 Has the effect of a basic rotation for tool axis **Z** **Further information:** "Basic rotation and 3D basic rotation", Page 235
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 218
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 218
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 218

Parameter	Meaning
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 218
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 218
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 218
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 218
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 218
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 218
ACTNO	Active workpiece preset The control automatically enters **1** in the active row.
LOCKED	Write-protection of the table row

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 235

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.

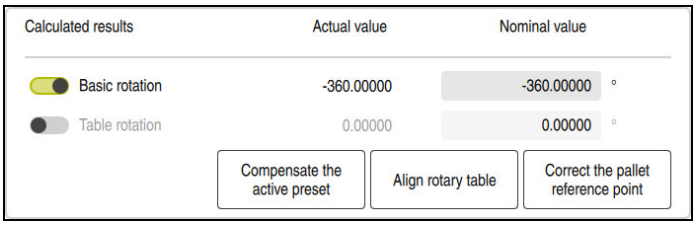
Further information: "Machine coordinate system M-CS", Page 218

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** touch probe function to determine workpiece misalignment. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating for the basic rotation of a workpiece", Page 401



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 524

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 524

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

19.11.1 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.

19.11.2 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

- Activate the **Edit** 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.

19.11.3 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 581

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 234
- 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

19.12 Tables for AFC (#45 / #2-31-1)

19.12.1 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 280

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 531

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
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.
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 288
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 288
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)
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)
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.
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)

Parameter	Meaning
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**
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.

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

19.12.2 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 528
- Setting up and using AFC

  Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 280

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 528

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

19.12.3 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 280

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
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

19.12.4 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 173
- Handwheel superimpositioning with GPS (#44 / #1-06-1)
  Further information: "The Handwheel superimp. function", Page 299
- 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 the **Manual** operating mode
  Further information: "Touch probe functions in the Manual operating mode", Page 387

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 538
HR 520, HR 520 FS HR 550 FS	Handwheel with display with wireless transmission	Page 541

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 127

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 536
- Handwheels with display
  Further information: "Handwheel with display", Page 541

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 540
3 Keys for the speed level
  Further information: "Speed levels", Page 539
4 Traverse direction keys
5 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **NC stop**
6 Handwheel permissive buttons
  Further information: "Activating and deactivating a handwheel", Page 539
7 Dial
8 **Emergency stop** key

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

As an alternative to the **Handwheel** key, you can also use the **Handwheel** toggle switch in the function bar of 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

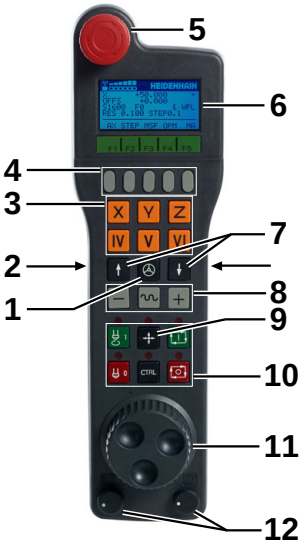
This chapter contains information specific to the HR 520, HR 520 FS and HR 550 FS handwheels with display. Handwheels with display can perform more functions than handwheels without display.

Related topics

- Overview of handwheels
  Further information: "Overview", Page 536
- Handwheels without display
  Further information: "Handwheel without display", Page 538

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 546

2 Handwheel permissive buttons on the sides

3 Axis keys

4 Soft-key selection keys
Further information: "Handwheel soft keys", Page 543

5 **Emergency stop** key

6 Display
Further information: "Contents of display", Page 542

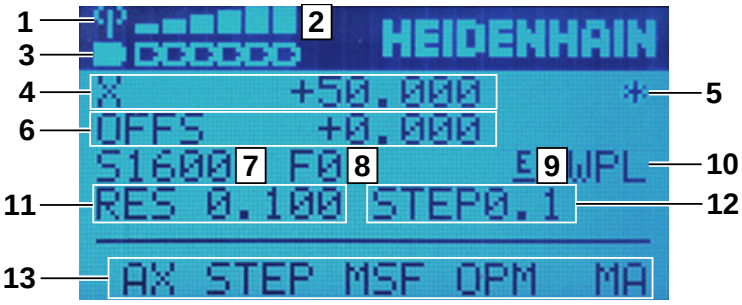
7 Speed level
Further information: "Speed levels", Page 544

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 540

10 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **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 299
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 242
11 Traverse per wheel detent stop
Further information: "Speed levels", Page 544
12 Incremental jog active or inactive, and jog increment
Further information: "Incremental jog positioning", Page 548
13 Handwheel soft keys
Further information: "Handwheel soft keys", Page 543

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 548
MSF	**F3**	Define cutting data, miscellaneous functions, and presets **Further information:** "Defining the spindle speed S", Page 547
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 548

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.

Always dock an HR 550 FS in its HRA 551 FS when not in use. This has the following benefits:

- The safety self-test can be performed:
- The NiMH rechargeable battery of the handwheel is always fully charged.
- An unexpected emergency stop reaction due to an empty NiMH rechargeable battery is avoided.
- Confusion between different wireless handwheels is avoided.

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 550

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

X

- 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 535
- HR 550 FS wireless handwheel
  Further information: "Special features of the HR 550 FS wireless handwheel", Page 545
- TNCdiag
  Further information: "TNCdiag", Page 643

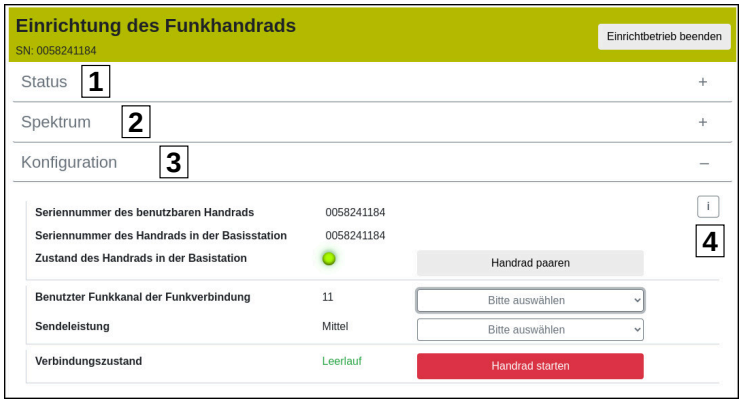
Requirements

- 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 recommended 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.

20.4.1 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 in **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
- Select the transmitter power in the **Transmitter power** selection menu (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.

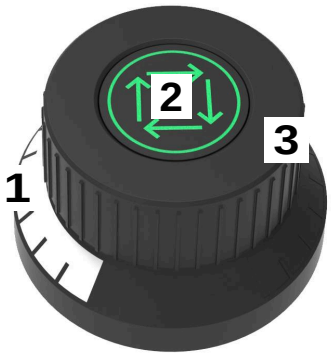
  Refer to your machine manual.
- Control is fully booted

  The control only detects the override controller once the machine control voltage has been acknowledged.
- Tool inspection has been performed

  Further information: Programming and Testing User's Manual

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 555

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

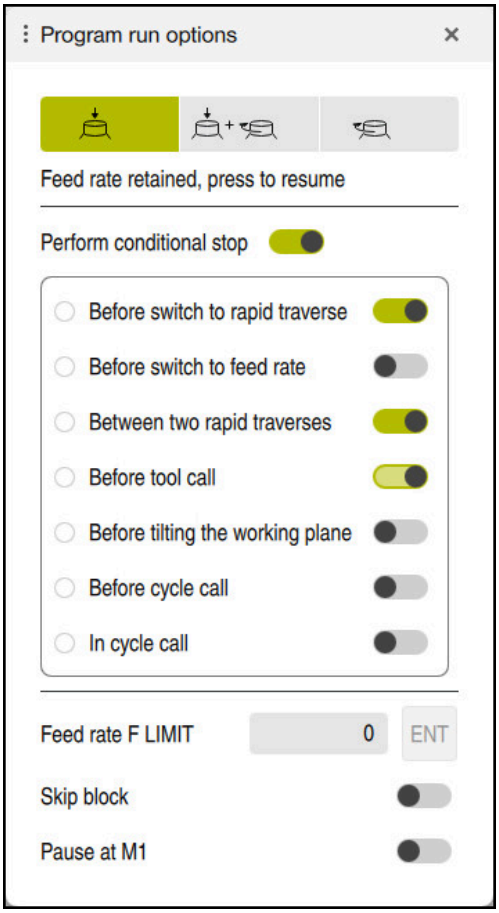
The **NC Start** button lights up green. The actual color may differ, depending on the machine.

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 428
- 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 557

The following functions are available also without the override controller:

- **Feed rate F LIMIT**
  Further information: "Feed rate limit F LIMIT", Page 431
- **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

The control displays active breakpoints on the **PGM** tab of the **Status** workspace.
Further information: "The PGM tab", Page 144

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 556

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

NOTICE

Caution: Possible material damage!

Incorrect operation may cause mechanical damage to the override controller, for example when using it as a handle for moving the operating panel.

- Use the override controller only for the following purposes:
  - 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

- 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 **Tests** column.

  Further information: Programming and Testing User's Manual
- 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 628
- Using Extended Workspace to operate a Windows PC through an additional connected monitor
  Further information: "Extended Workspace", Page 566

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**.

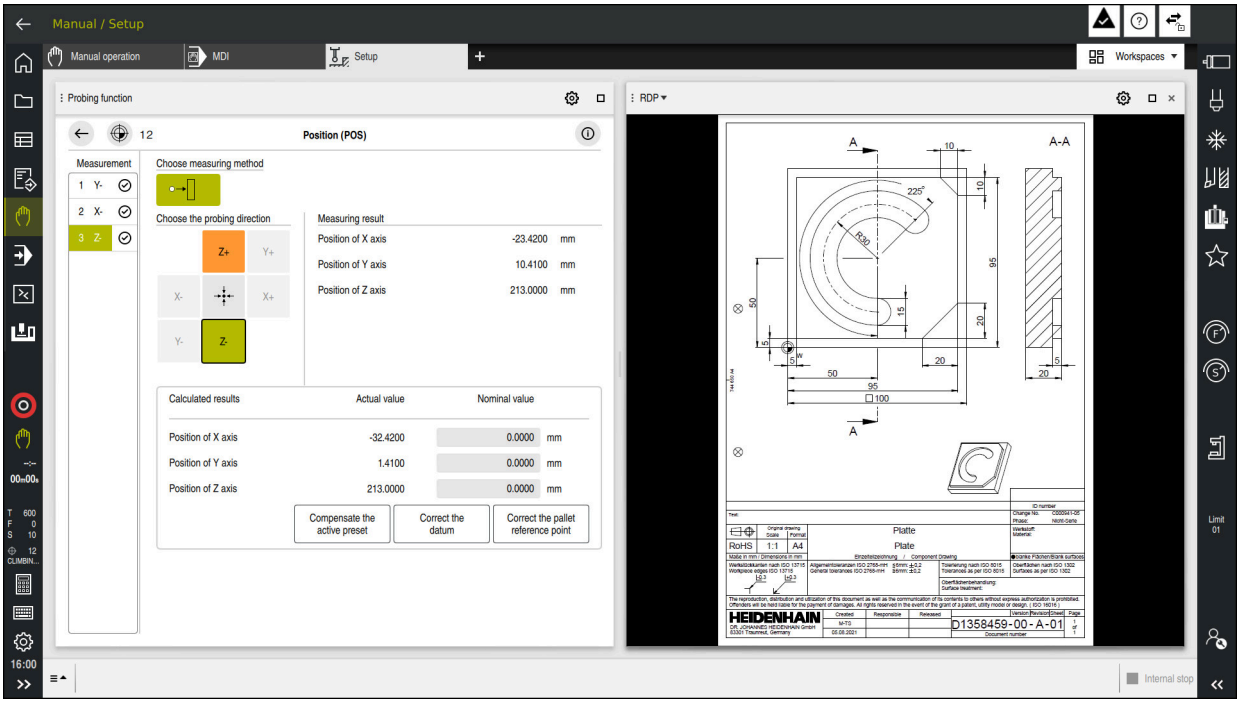
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 629

If you open Embedded Workspace as an operating mode, the control displays a full-screen version of the Windows PC user interface in it.

If you open Embedded Workspace as a workspace, you can change the size and position of the workspace as you wish. The control rescales the user interface of the Windows PC after each modification.

Further information: "Workspaces", Page 92



Embedded Workspace as workspace with opened PDF file

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 in 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 564
- ITC hardware expansion

  Further information: "Hardware enhancements", Page 86

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 685
- 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 628

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 571

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).

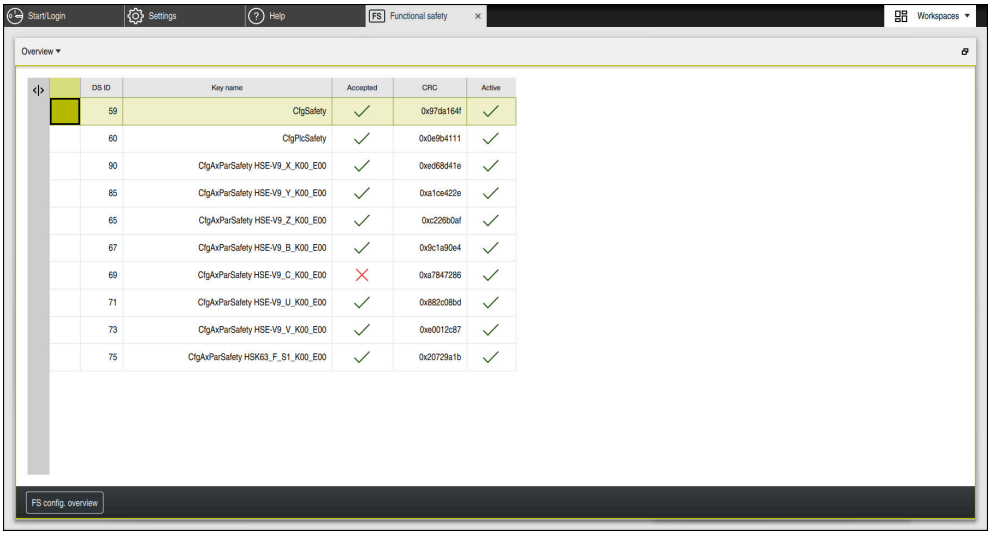
Further information: "Axis display and position display", Page 128

The Functional safety application

Refer to your machine manual.

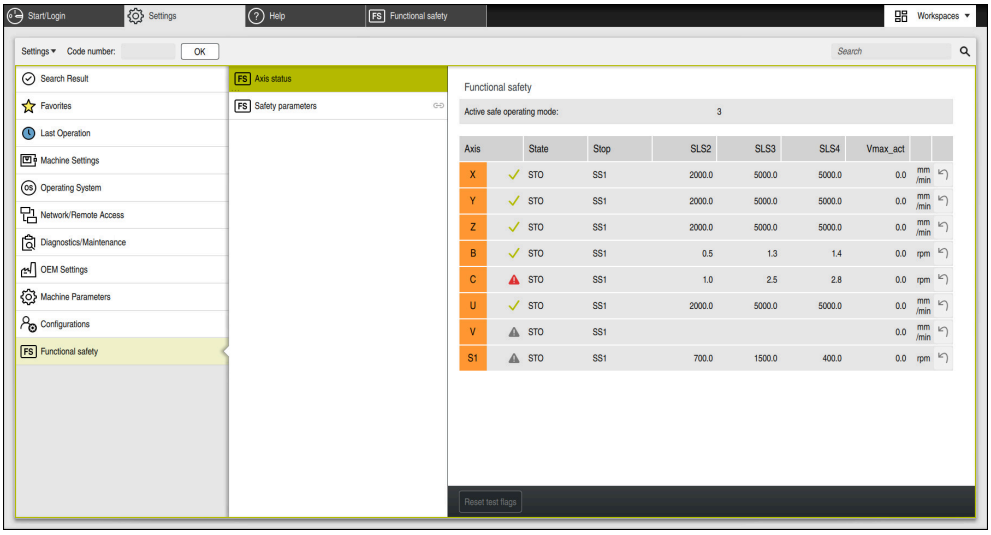
The machine manufacturer configures the safety functions in this application.

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 570
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 655

  Further information: "User administration roles and rights", Page 709

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 573

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 573
⚠	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 the **Move to ref. point** application
- Select **Axis position** 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 **Axis position** 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 581
			General Information **Further information:** "The General Information menu item", Page 584
			SIK **Further information:** "The SIK menu item", Page 586
			Machine Times **Further information:** "The Machine Times menu item", Page 589
			Overview of touch probes **Further information:** "Overview of touch probes menu item", Page 590
			Adjustment of analog voltage offset **Further information:** "The Adjustment of analog voltage offset menu item", Page 593
			Set Up Wireless Handwheel **Further information:** "Setting up a wireless handwheel ", Page 550
	Operating System		**Date/Time** **Further information:** "The Adjust system time window", Page 594
			Language/Keyboards **Further information:** "Conversational language of the control", Page 595
			About HeROS **Further information:** "Information on licensing and use", Page 81
			SELinux **Further information:** "SELinux security software", Page 597
			UserAdmin **Further information:** "The User administration window", Page 664
			Current User **Further information:** "The Active user window", Page 665
			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 598
			**Network** **Further information:** "Ethernet interface", Page 602
			**PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "PKI Admin", Page 609
			**OPC UA** **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 611
			**DNC** **Further information:** "The DNC menu item", Page 619
			**Embedded Workspace** Show the connection status **Further information:** "Embedded Workspace (#133 / #3-01-1)", Page 564
			**Printer** **Further information:** "Printers", Page 622
			**VNC** **Further information:** "The VNC menu item", Page 625
			**Remote Desktop Manager** **Further information:** "The Remote Desktop Manager window (#133 / #3-01-1)", Page 628
			**Real VNC Viewer** Connect to a remote device via a VNC server Available only to network specialists
			**Firewall** **Further information:** "Firewall", Page 634

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 639
			perf2 Check processor load and process load
			TNCdiag **Further information:** "TNCdiag", Page 643
			TNCscope Available only to authorized specialists
			NC/PLC Backup **Further information:** "Backup and restore", Page 640
			NC/PLC Restore **Further information:** "Backup and restore", Page 640
			Touchscreen Cleaning The control disables the touchscreen for input for 90 seconds.
			Update the documentation **Further information:** "Update the documentation", Page 644
			Expanded logging If this function is active, the control will save graphics journal data. These data may be needed by the HEIDENHAIN Customer Service for diagnostic purposes. If you activate this function, you need to restart the control. Deactivate this function after the fault has been fixed to reduce the amount of memory used.
	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 646
	Configurations		**Configurations** **Further information:** "Configuring the control's user interface", Page 653

Icon	Category	Icon	Menu item
FS	**Functional safety**	FS	**Axis status** **Further information:** "The Axis status menu item", Page 571
		FS	**Safety parameters** **Further information:** "The Functional safety application", Page 570

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 646
	When this code number is active, the control displays a different icon for the **Home** operating mode.
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.

The Machine and Simulation areas

Channel Settings

You can define the following settings:

Setting	Meaning
Active Kinematics	Use the **Active Kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, have been programmed for other machines. The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose. The control displays the active kinematics model in the **Machine** mode of the **Simulation** workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage. **Further information:** "Tool usage test", Page 212 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.

Oper. station for program interruption

The control displays this function only in the **Machine** area.

Refer to your machine manual.
The machine manufacturer uses the optional machine parameter **userControl** (no. 144101) to define if this area is available.

You can define the following settings for the **Manual traverse** during program run:

Setting	Meaning
Standard operating station	You select the operating element to be used for moving the axes: ■ **MB** machine operating panel ■ **HR** handwheel
Remember last operating station	If the toggle switch is active, the control will not consider the selected **Standard operating station**. The control remembers the state of the **Handwheel** toggle switch from the last manual traverse, and activates the associated operating element.

Further information: "Manual traverse during an interruption", Page 437

Traverse Limits

The control displays this function only in the **Machine** area.

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 514
- 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 516

  Further information: "Tooling list (#93 / #2-03-1)", Page 518

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. If, for example, the HEIDENHAIN service department requires the NC software number, you will find that number in this area.

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 ■ **NC-SW** Number of the NC software ■ **NCK** Version of the software Is managed by HEIDENHAIN
PLC	**PLC-SW** Number or name of the PLC software Is managed by the machine manufacturer
Operating System	■ **HEROS version** ■ **Yocto version** Is managed by HEIDENHAIN

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

If the machine manufacturer has defined a version number for at least one OEM cycle, the control displays this information in the **OEM cycles** sub-area.

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 612

The Machine information area

Sub-area	Meaning
Hardware	**RAM** RAM of the main computer

The control also 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 612

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 73

Description of function

To navigate to this function:

Settings ▶ **Machine Settings** ▶ **SIK**

The **SIK** menu item consists of the following areas:

Area	Contents
SIK Information	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**
Machine manufacturer key	The machine manufacturer can define a manufacturer-specific password for the control.
General key Only with **SIK**	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: ■ **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.
Enabling of options Only with **SIK2**	You can select a key file in order to enable software options. You can obtain the key file from your machine manufacturer. **Further information:** "Enabling software options", Page 588
Software Options	The control displays all available options in a table. **Further information:** "The Software Options area", Page 587

The Software Options area

The table with the available software options contains the following columns:

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**.

24.5.2 Enabling software options

Control equipped with SIK

To enable a software option for a control with **SIK**, you need an enabling code (from the machine manufacturer, for example).

To enable a software option with **SIK**:

- ▶ Switch on the power supply of the control and the machine
- > The control starts the operating system and displays the **Power interrupted** message.
- ▶ Navigate to the **SIK** menu item
- ▶ Scroll to the **Software Options** area

Set

- ▶ Select **Set** in the **Option** column
- > The control opens the **Unlock the option** window.
- ▶ Select the desired software option in the selection menu as needed
- ▶ Enter the activation code

Set

- ▶ Select **Set**
- > The control restarts and activates the software option.

Control equipped with SIK2

To enable a software option for a control with **SIK2**, you need a key file (from the machine manufacturer, for example).

To enable a software option with **SIK2**:

- ▶ Switch on the power supply of the control and the machine
- > The control starts the operating system and displays the **Power interrupted** message.
- ▶ Navigate to the **SIK** menu item

Select

- ▶ In the **Enabling of options** area, select the **Select** button
- > The control opens the **Select key file for options (SIK2_xxx.txt)** window.
- ▶ Navigate to the folder containing the key file

Open

- ▶ Select **Open**
- > The control reads the key file and opens the **Restart** window.

Restart

- ▶ Select **Restart**
- > The control restarts and activates the software option.

If a key file is present in the **TNC:\SIK2** folder, the control will read that file automatically during the next startup.

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 594

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 506
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)
  Further information: "Machine parameters", Page 646

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 643
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 The 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.

Only use this function after consultation with your machine manufacturer.

Requirement

- Machine with analog axes
- If user administration is active, the NC.Setter role
  Further information: "List of roles", Page 709

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Adjustment of analog voltage offset

If there are any analog axes, the control displays a table with the following columns:

Column	Meaning
Axis	All analog axes
In 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.
Current offset	Currently stored voltage offset
New offset	Currently ascertained voltage offset If permitted by the axis configuration, you can adjust the voltage offset here.
Deviation	Deviation of the ascertained voltage offset from the stored offset

If you adjust the voltage offsets in the **New offset** 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 589

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 646

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 596
  - **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 634

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 602

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 600
Remove	Delete an existing connection
Copy	Copy connection **Further information:** "The Mount assistant window", Page 600
Edit	Edit the connection settings **Further information:** "The Mount assistant window", Page 600
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 601
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

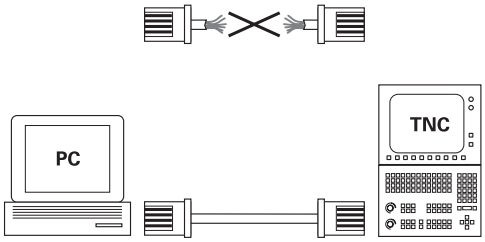
- Firewall settings
  Further information: "Firewall", Page 634
- Network drives on the control
  Further information: "Network drives on the control", Page 598
- External access
  Further information: "The DNC menu item", Page 619

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 688 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.

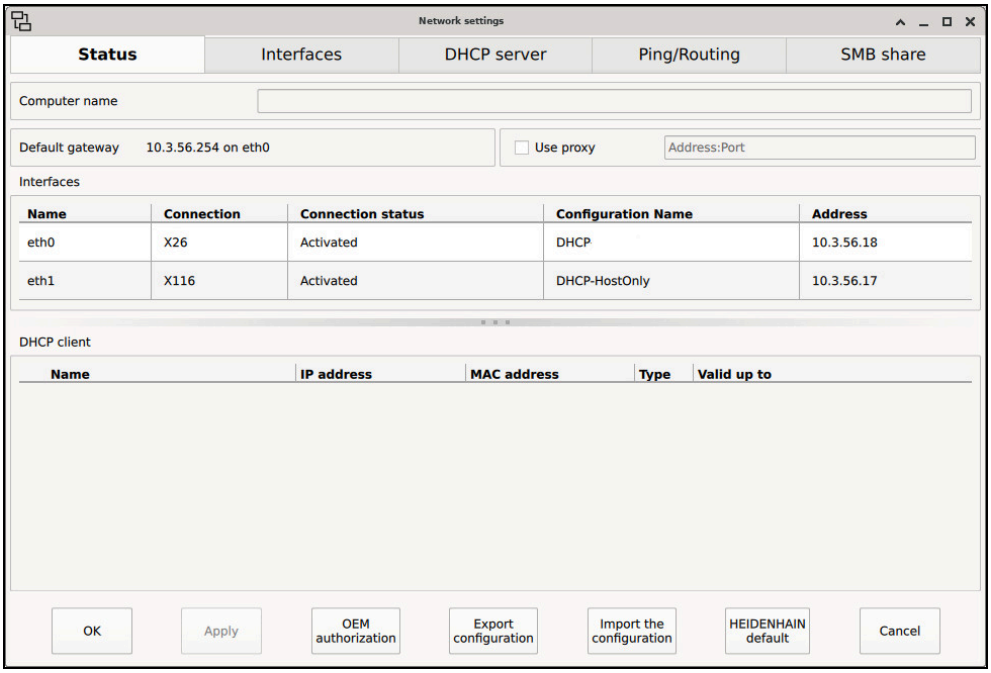
Related topics

- Network configuration
  Further information: "Network configuration with Advanced Network Configuration", Page 702
- Firewall settings
  Further information: "Firewall", Page 634
- Network drives on the control
  Further information: "Network drives on the control", Page 598

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 608
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 606
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 702 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 702

- 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 666

  You must also regenerate the server certificates of **OPC UA NC Server** (#56-61 / #3-02-1*).

  Further information: "Login options", Page 613
- 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 595

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 617

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 613 ■ **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 an **OPC UA** connection (#56-61 / #3-02-1*), you also need to assign an **OPC UA** license to the certificate. **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 618
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 617
- User roles and user rights for OPC UA
  Further information: "User administration roles and rights", Page 709
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 699

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 634
- 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

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 205
- Validate 3D models for tools (#140 / #5-03-2)

  Further information: "Tool model (#140 / #5-03-2)", Page 209

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 585
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)

  Further information: "The Info about machine manufacturer area", Page 584

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 609

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 616

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 655

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 616
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 617
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 618
PKI Admin	Open the **Administration of the PKI Infrastructure** window **Further information:** "PKI Admin", Page 609
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 619

When a connection is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 105

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 709

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 #3-02-10 using the **OPC UA NC Server - License Settings** window
  Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 618
- Managing certificates with the **PKI Admin** menu
  Further information: "PKI Admin", Page 609

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 616

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 an 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 #3-02-10.

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 617
- Managing certificates with **PKI Admin**

  Further information: "PKI Admin", Page 609

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 598
- Setting up a network
  Further information: "Ethernet interface", Page 602
- TNCremo
  Further information: "PC software for data transfer", Page 692
- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 628

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 105

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 678

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 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.

Overview of ports for DNC connections

A DNC connection can use the following protocols:

- SSH (secure connection) with RPC-Secure or LSV2-Secure
- RPC
- LSV2

> 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.
>
> **Further information:** "SSH-secured DNC connection", Page 678

Depending on the protocol being used, the connection is established via the following ports:

Port	Control or client PC	Protocol	Firewall
22	Control and client PC with secure connection	RPC-Secure LSV2-Secure	Allow **SSH**
19000	Control	LSV2	Allow **DNC**
19003	Control	RPC	Allow **DNC**
19010-19034	Client PC	RPC	Allow port range

For the client PC, an RPC-Secure connection via an SSH tunnel also uses port 19036. You don't need to allow port 19036, since the secure connection is established via port 22.

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 PostScript emulation in the printer menu.
  Further information: "Note", Page 624

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 624

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

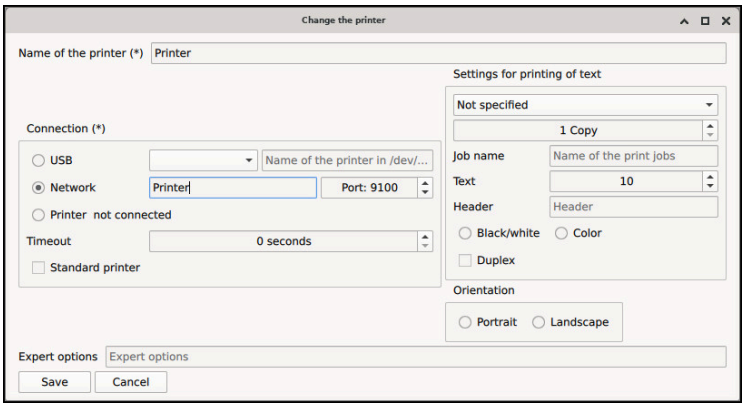
Icons and buttons

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

Icon or button	Meaning
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.

Area	Meaning

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 634
- Remote Desktop Manager (#133 / #3-01-1)
  Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 628

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 627

Further information: "Icons on the control's user interface", Page 105

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 634

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 619

Requirements

- Remote Desk. Manager (#133 / #3-01-1) software option
- Existing network connection
  Further information: "Ethernet interface", Page 602

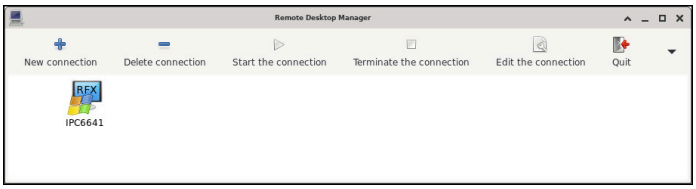
Description of function

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 629
- **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control
  Further information: "VNC", Page 629
- **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 632
Delete connection	Delete the selected connection
Start the connection	Start the selected connection **Further information:** "Establishing and starting a connection", Page 632
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 630
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection **Further information:** "Exporting and importing connections", Page 633
Export the connections	Back-up the selected connection **Further information:** "Exporting and importing connections", Page 633

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 632

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 630
- 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 **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.

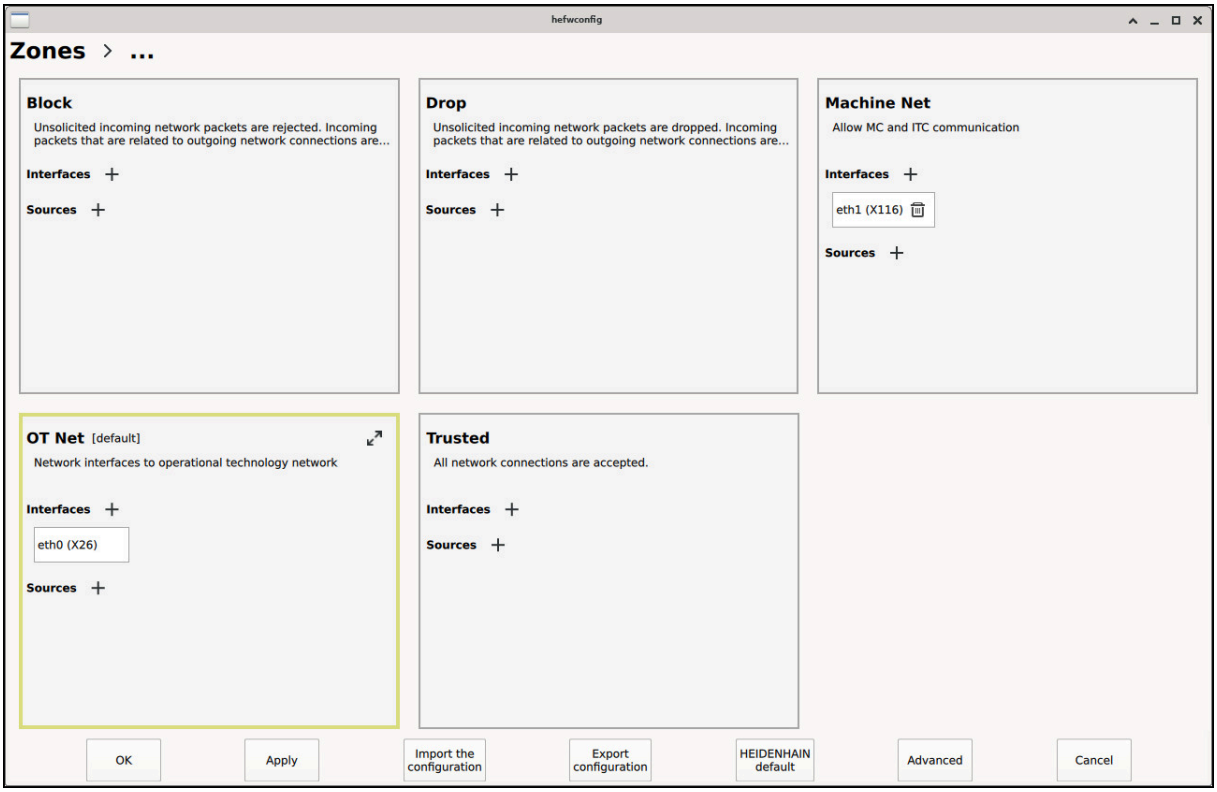
Related topics

- Existing network connection
  Further information: "Ethernet interface", Page 602
- SELinux security software
  Further information: "SELinux security software", Page 597
- Ports required for a DNC connection
  Further information: "Overview of ports for DNC connections", Page 621
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 699

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Firewall



Overview of zones

The **Firewall** 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 636

You can edit the configuration of the zones.

Further information: "Settings of the zones", Page 637

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 **Firewall** 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
Logging	Activate or deactivate logging of rejected packets in the service file **Further information:** "Service file", Page 381 This information might be needed by HEIDENHAIN Customer Service for diagnostic purposes. Deactivate this function after the fault has been fixed, so as not to impair the performance.
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 If the machine manufacturer has stored standard values, the control displays the **OEM default** button
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 637

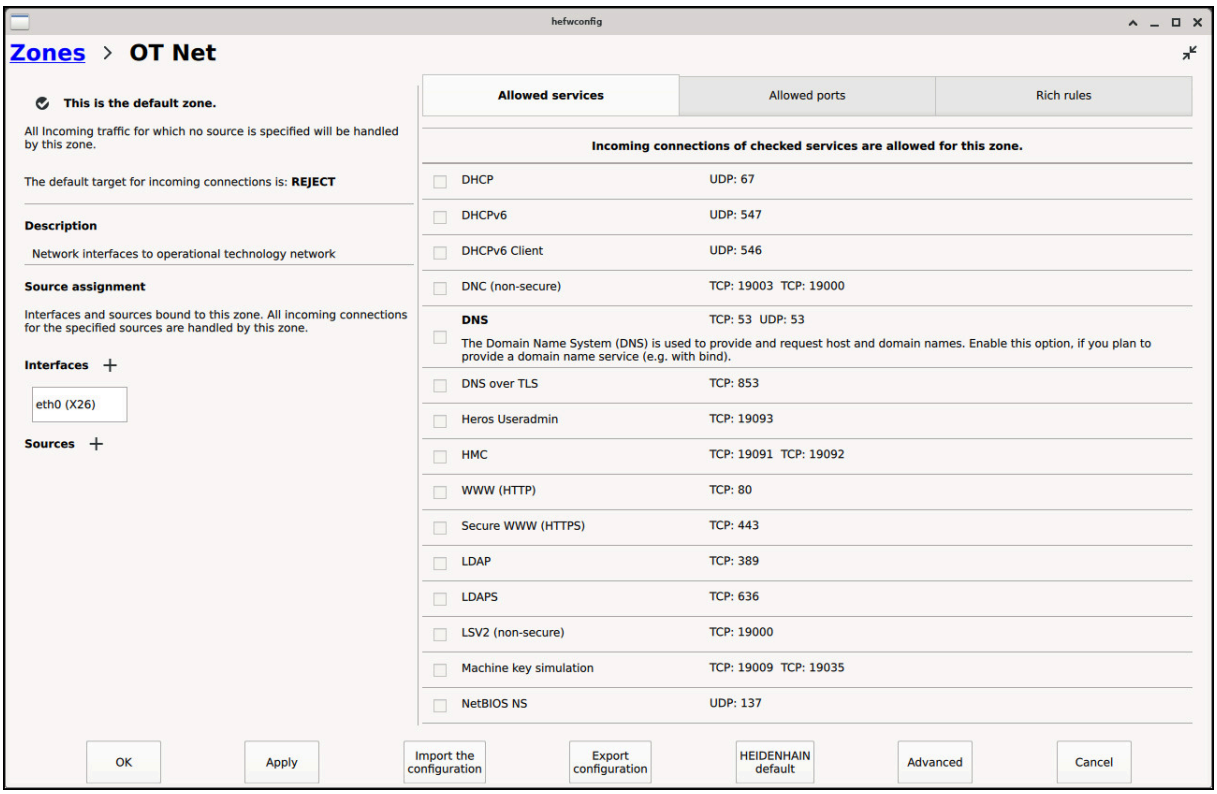
Zones

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.
PNC Net	This zone is the default zone on a PNC7. Default target: **REJECT** with exceptions This zone accepts the services **SSH** and **VNC**. On a PNC 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 634
- Network settings
  Further information: "Network configuration with Advanced Network Configuration", Page 702

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 choose 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 642
- 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 642
- 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 550

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 58
- Product aid on the HEIDENHAIN website
  https://content.heidenhain.de/doku/tnc_guide/html/en/

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 645

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).

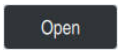
24.24.1 Transferring TNCguide

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 580
- 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.

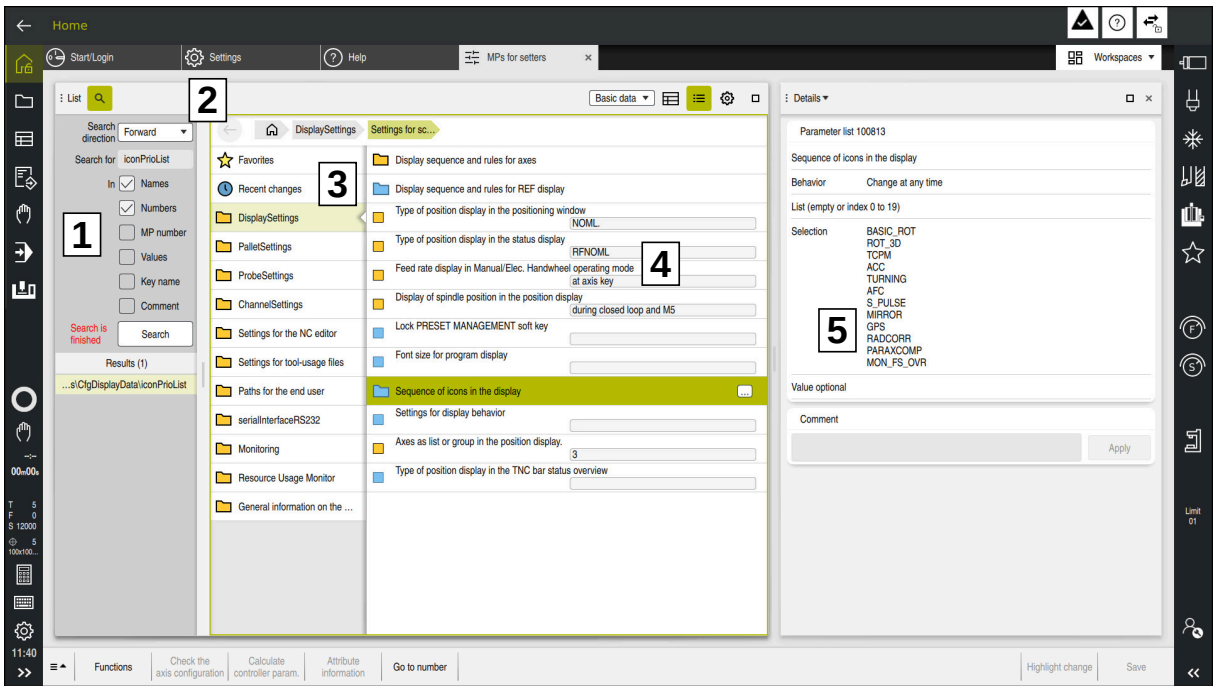
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **Details**
- **Document**
- **List**

You cannot close the **List** workspace.

The configuration editor areas



MPs for setters application with selected machine parameter

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 650
- Open or close the **Details** workspace

  Further information: "The Details workspace", Page 651
- Open or close the **Configuration** window

  Further information: "The Configuration window", Page 651

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 651

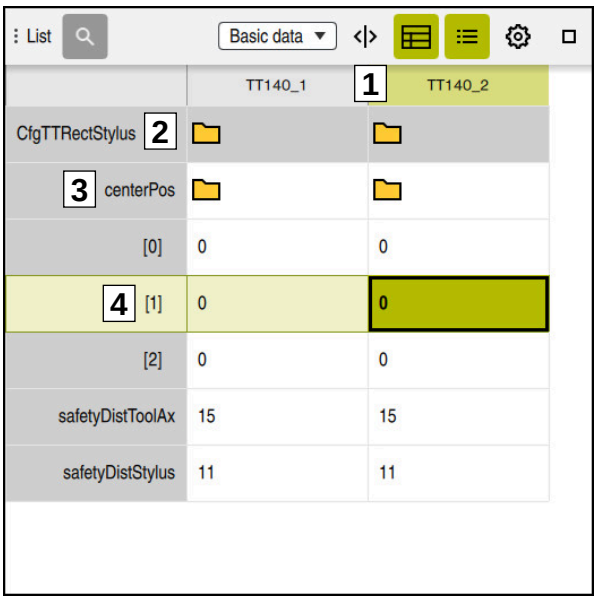
Icons, buttons and shortcuts

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 650
	Active or deactivate the **Change column width** window 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 651
	Open or close the **Configuration** window **Further information:** "The Configuration window", Page 651
	**Expand**
	**Collapse** Only if the **Display in tree view** toggle switch is active **Further information:** "The Configuration window", Page 651
	Select **Recent changes**
	Object exists - Data object - Directory - Parameter list
	Object empty
	**Open context menu** **Further information:** Programming and Testing User's Manual
	Machine parameter exists

Icon, button, or shortcut	Meaning
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter not readable and not editable
	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** 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.
Highlight change	The control marks the selected machine parameter as changed, even though the value remains the same. For machine manufacturers only
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.
- **Do not show attributes without values**

  If the toggle switch is active, the control hides optional machine parameters and parameter lists.
- **Do not show empty folders**

  If the toggle switch is active, the control hides empty data objects and data-object lists.

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 **Home** operating mode icon shown in the TNC bar changes if the code number 123 is enabled. The icon additionally shows an unlocked padlock to illustrate that machine parameters can be accessed.
- 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 655
- 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 92
- Control interface
  Further information: "Areas of the control's user interface", Page 88

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 108
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.
Enable autosave	If the toggle switch is active, the control automatically saves changes to the active configuration. If the toggle switch is not active, you can save the active configuration with the **Save current settings** function. If, for example, more than one person is working on the same configuration, the configuration will not be overwritten unintentionally.

The control displays the following information about all available configurations in a table:

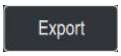
Column	Meaning
Configuration Name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 654
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**.

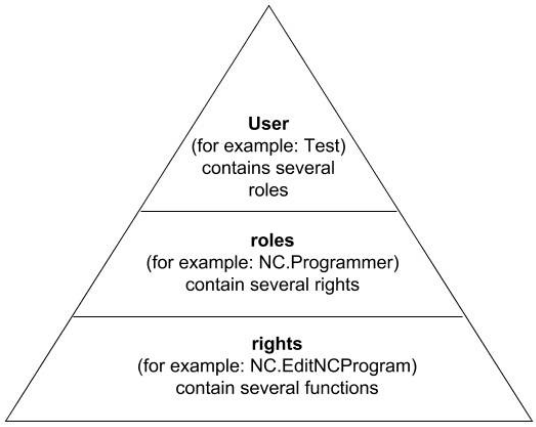
Description of function

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 656
- Roles
  Further information: "Roles", Page 658
- Rights
  Further information: "Rights", Page 658



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 662

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 663

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 665

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 709

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 678

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 712

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 666

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 **UserAdmin**
- > The control opens the **User administration** window in the **Settings** tab.

  Further information: "The User administration window", Page 664

Activating user administration

To activate user administration:

- Select **User administration active**
- > The control shows the message **Enter password for user 'useradmin'**.
- 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 656

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 659

- Repeat the password
- Select **Set new password**
- > The control shows the message **Settings 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 **OK**

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 666

Creating a new user

To create a new user:

- Select the **User administration** tab
- Select the **Editing on** checkbox
> The control opens the **Enter password of LDAP database** window.
- Enter password
- Select **OK**
- 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 658

- Select **Close**
> The control closes the **Add role** window.
- Select **Apply**
> The control adopts the changes.
- Select **Exit**
> 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 656

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 **Exit**
- > 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 685
- 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 628

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 665

Requirement

- If user administration is active, the HEROS.Admin role
  Further information: "List of roles", Page 709

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 660
User administration	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 662
Password settings	Define password requirements **Further information:** "Password settings", Page 659

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 678
- Logon with smartcards
  Further information: "Logon with smartcards", Page 674
- Available roles and rights
  Further information: "User administration roles and rights", Page 709

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 roles and 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 664
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 678 **Further information:** "OPC UA NC Server (#56-61 / #3-02-1*)", Page 611
Create token	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 674
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 659

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 656

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 666
- **LDAP on remote computer**
  Further information: "LDAP database on a remote computer", Page 667
- **Connection to Windows domain**
  Further information: "Connection to Windows domain", Page 668

> 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 667
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 668

Requirements

- User administration is active
  Further information: "Activating user administration", Page 661
- **useradmin** user is logged on
  Further information: "Users", Page 656

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 666
- Connecting a Windows domain with user administration

  Further information: "Connection to Windows domain", Page 668

Requirements

- User administration is active

  Further information: "Activating user administration", Page 661
- **useradmin** user is logged on

  Further information: "Users", Page 656
- 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 668

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 **Open**
- 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 666
- Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 667

Requirements

- User administration is active
  Further information: "Activating user administration", Page 661
- **useradmin** user is logged on
  Further information: "Users", Page 656
- 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 672

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 670
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 672
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 672
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 671

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:

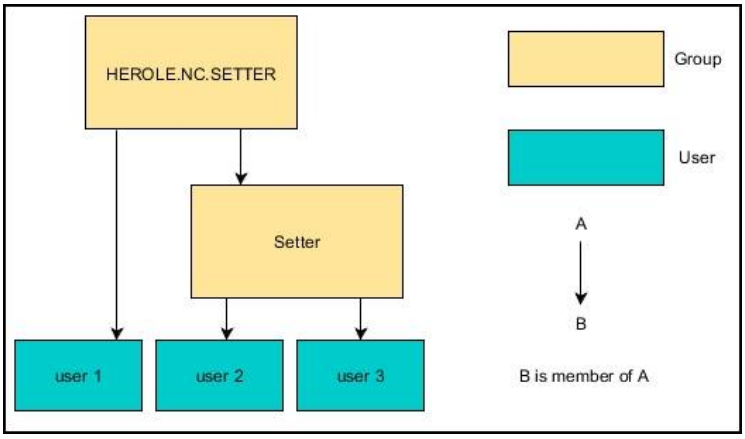
- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

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.

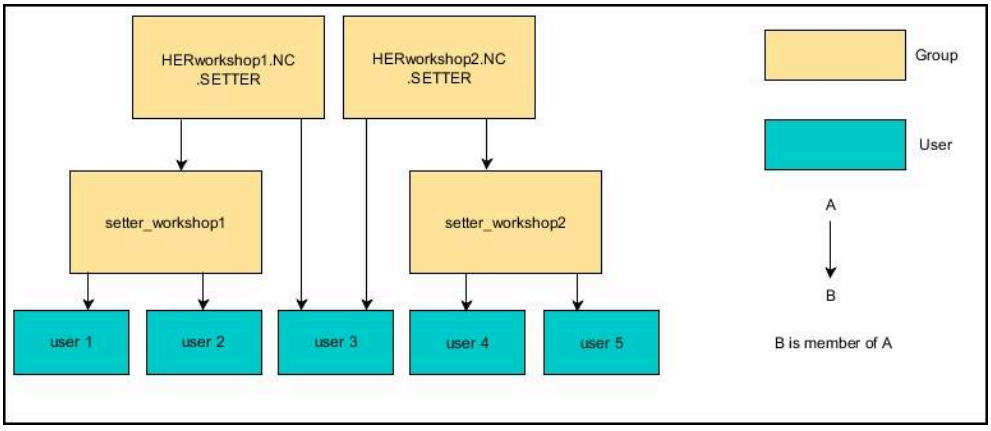
Example 1

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 with 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 674
- Configuring user administration
  Further information: "Configuring user administration", Page 660

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 664

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 677

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 with user administration", Page 673

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 675

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 685

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 675

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.

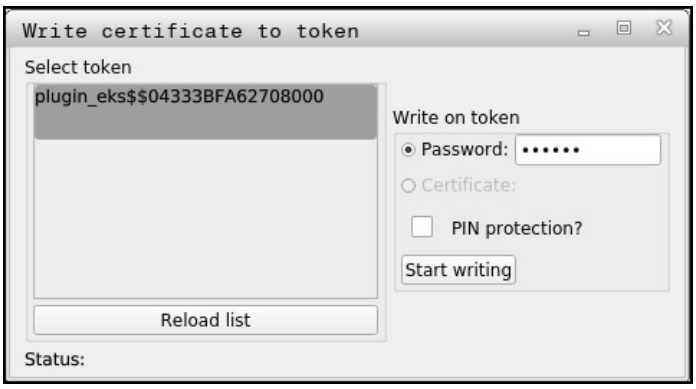
- > The control opens a window with the message **Password expired. Change the password now.**
- Enter the current password
- Enter a new password
- Repeat the new password
- > The control uses the new user to log you in.
- > The control displays this user in the dialog during the next logon procedure.

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
- Activate the **PIN protection?** check box, if desired
- 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.

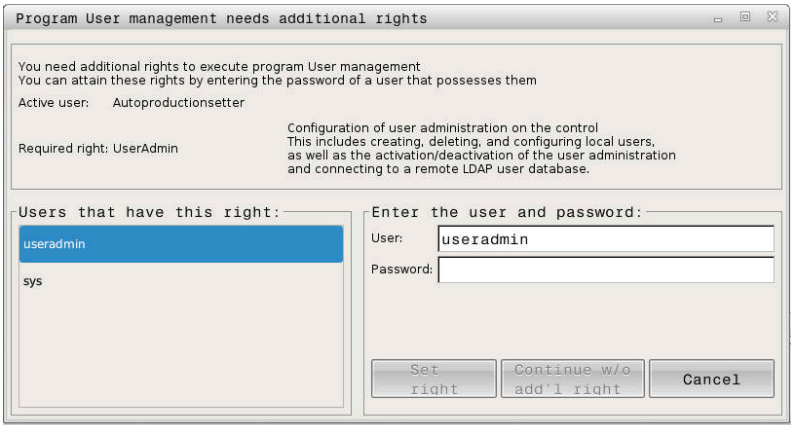
Related topics

- Temporarily granting additional rights in the **Active user** window
  Further information: "The Active user window", Page 665

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 634
- Roles for remote logon
  Further information: "Roles", Page 658

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

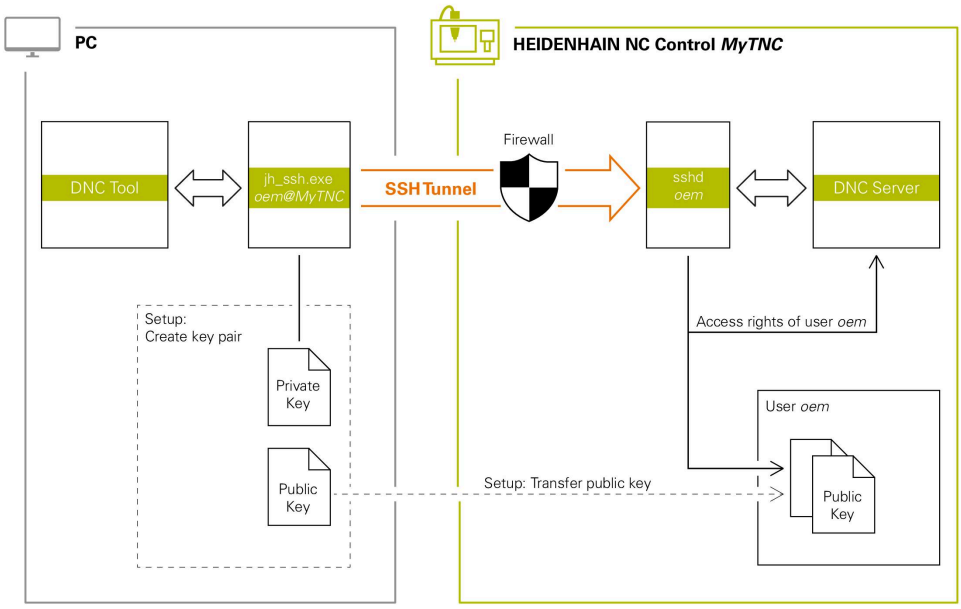
Description of function

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 611
- 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.

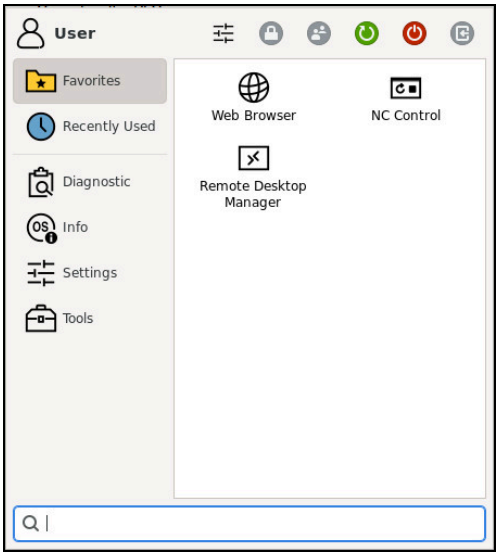
Related topics

- Opening HEROS functions through the **Settings** application
  Further information: "The Settings application", Page 575

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 688



Standard view of the HEROS menu

The HEROS menu contains the following functions:

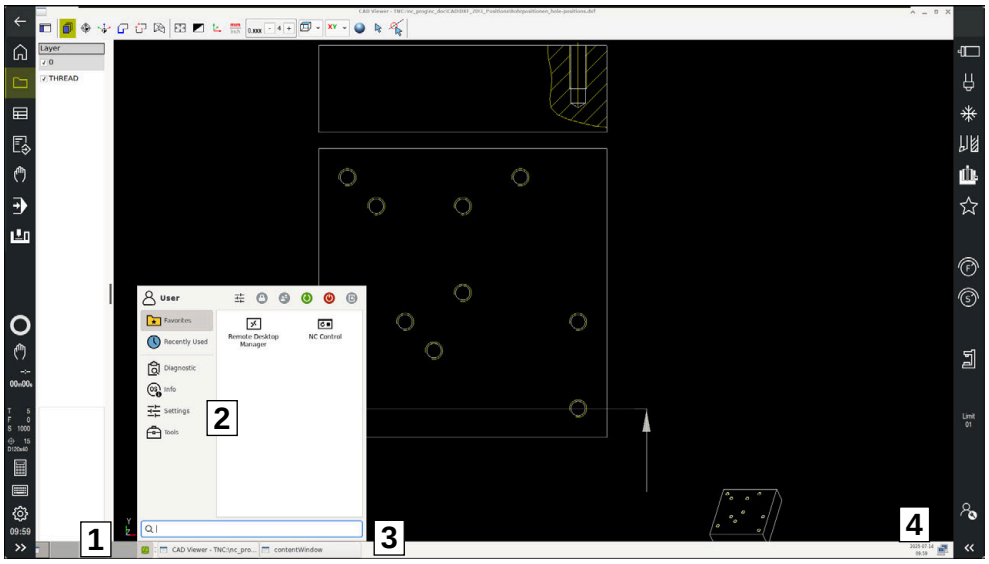
Area	Function
Header	■ User name **Further information:** "The Active user window", Page 665 ■ User-specific settings ■ Lock display Only if user administration is active ■ Switch users Only if user administration is active ■ Restart ■ Shut down ■ Log out Only if user administration is active **Further information:** "User administration", Page 655

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 639 ■ **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 643 ■ **TNCscope**: Available only to authorized specialists
Settings	■ **Adjust screen brightness**: Adjust screen brightness ■ **Screensaver**: Screensaver ■ **Current User** **Further information:** "The Active user window", Page 665 ■ **Date/Time** **Further information:** "The Adjust system time window", Page 594 ■ **Firewall** **Further information:** "Firewall", Page 634 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 595 ■ **Network** **Further information:** "Ethernet interface", Page 602 ■ **OEM Function Users** **Further information:** "Users", Page 656 ■ **OPC UA NC Server Connection Assistant** **Further information:** "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 617 ■ **OPC UA NC Server License** **Further information:** "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 618 ■ **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 611 ■ **Printer** **Further information:** "Printers", Page 622 ■ **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).

Area	Function
	■ **SELinux** **Further information:** "SELinux security software", Page 597 ■ **Shares** **Further information:** "Network drives on the control", Page 598 ■ **Touchscreen Calibration** ■ **Touchscreen Configuration** ■ **UserAdmin** **Further information:** "The User administration window", Page 664 ■ **VNC** **Further information:** "The VNC menu item", Page 625 ■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 689
Info	■ **About HeROS**: Open information about the operating system of the control ■ **About Xfce**: Open information on the Window manager
Tools	■ **Switch-off**: Shut-down or restart ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Document Viewer**: Display and print files (e.g., PDF files) ■ **Geeqie**: Open, manage, and print graphics ■ **Gnumeric**: Open, edit, and print tables ■ **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 ■ **Meld**: Compare 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 640 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 640 ■ **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 628 ■ **Ristretto Image Viewer**: Open graphics ■ **Secure Remote Access** **Further information:** "Secure Remote Access", Page 696

Area	Function
	■ **Combine fixtures** **Further information:** "Combining fixtures in the New Fixture window", Page 276 ■ **Virtual keyboard** ■ **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 685

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 634
- Network status

  Further information: "Ethernet interface", Page 602
- 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 88

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 88
- 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 646

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 692

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

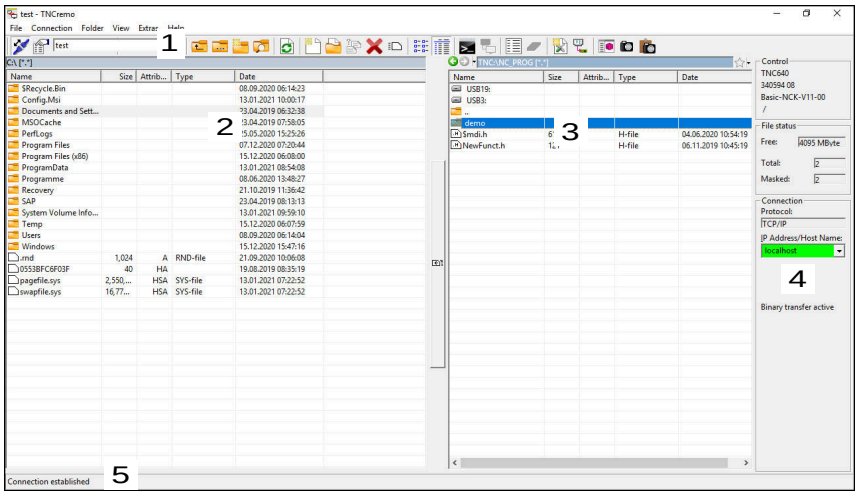
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 699

Requirements

- PC operating system:
  - Windows 10
  - Windows 11
- 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 the 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 655
- Principle of the SSH connection
  Further information: "Concept of transmission through an SSH tunnel", Page 679
- Firewall settings
  Further information: "Firewall", Page 634
- Comparison of the transmission duration of different protocols
  Further information: "Example: Transmission duration of different transmission types", Page 699

Requirements

- PC software TNCremo with version 3.3 or higher is installed
  Further information: "PC software for data transfer", Page 692
- **SSH** service is permitted in the firewall of the control
  Further information: "Firewall", Page 634

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 625

Requirements

- Existing Internet connection
  Further information: "Network configuration with Advanced Network Configuration", Page 702
- 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 625
- 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.

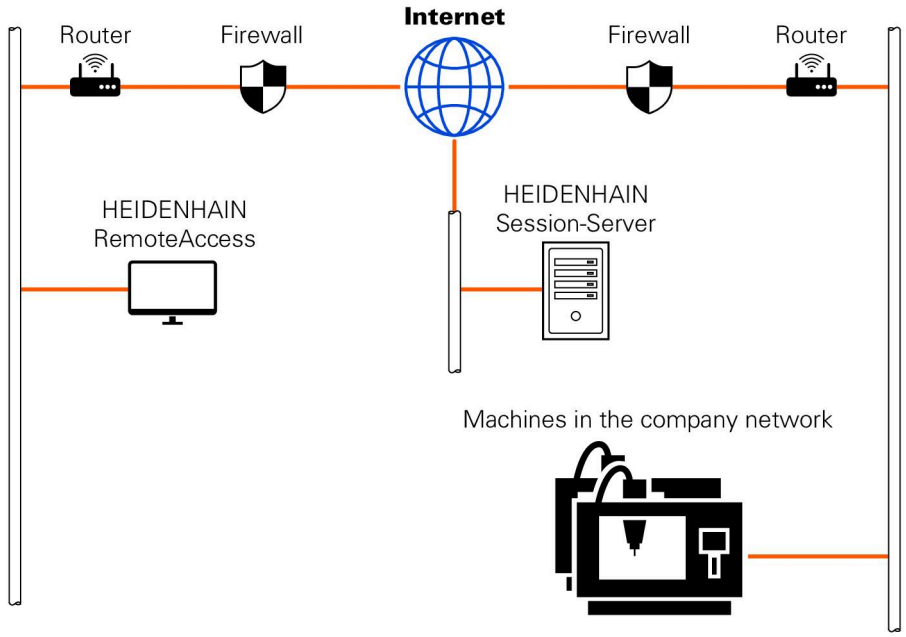
Description of function

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 a 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
Mount	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 640

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 692
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 598
- 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 from the control	Duration of transmission to the control	Further information
TNCremo v3	Approx. 3 min 30 s	Approx. 14 min 30 s	Page 692
TNCremo v4	Approx. 0 min 12 s	Approx. 0 min 12 s	
HEIDENHAIN DNC	Approx. 0 min 12 s	Approx. 0 min 12 s	Page 619
OPC UA NC Server	Approx. 0 min 27 s	Approx. 0 min 50 s	Page 611
SFTP	Approx. 0 min 09 s	Approx. 0 min 10 s	Page 694
SMB	Approx. 0 min 12 s	Approx. 0 min 09 s	Page 602

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: "CAD Viewer ", Page 349

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 276
CFT	ToolHolderWizard **Further information:** "Customizing tool carrier templates with ToolHolderWizard", Page 207
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 597

26.9 Network configuration with Advanced Network Configuration

Application

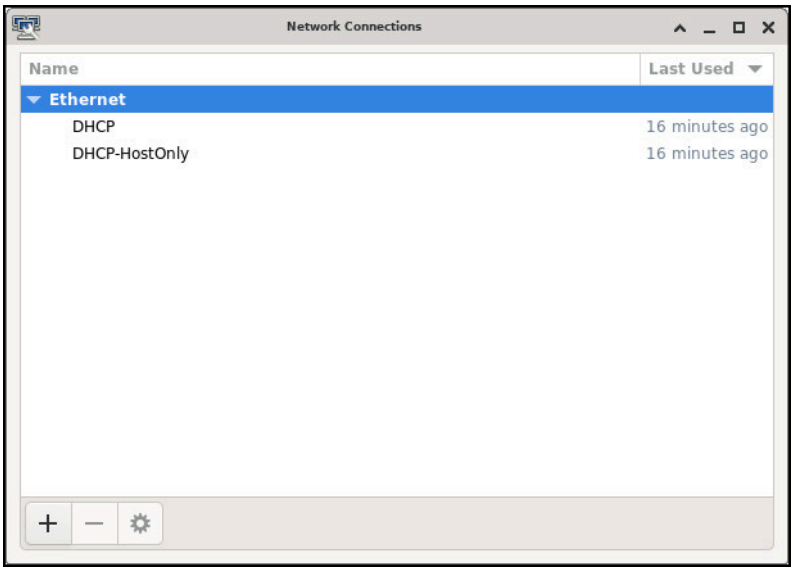
Use **Advanced Network Configuration** to edit or remove profiles for the network connection.

Related topics

- Network settings
  Further information: "The Editing network connection window", Page 703

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

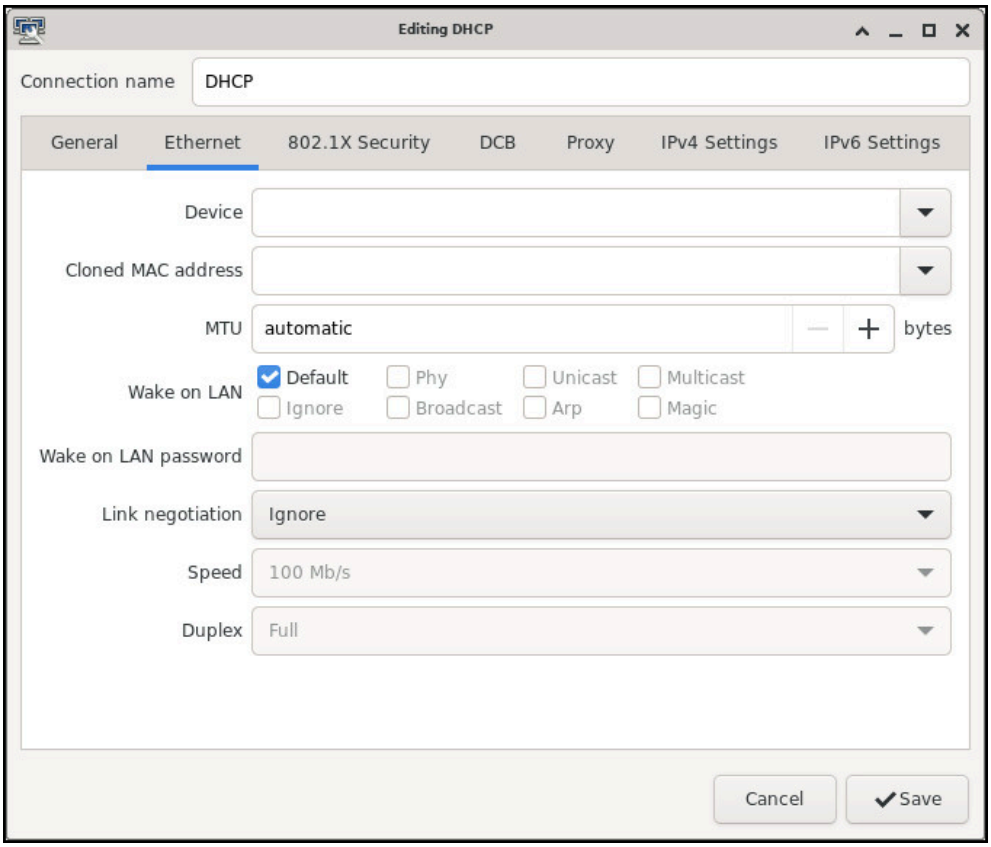
Icons in the Network Connections window

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 703

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 658

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.VMSharedFolders	■ vboxsf	■ 1000
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.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 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.UserAdmin	■ useradmin	■ 339
	■ HEROS.BackupMachine	■ backup	■ 338

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.LegacyUser	**Legacy-User** is the default role if user administration is inactive. HEIDENHAIN recommends assigning other roles to users when user administration is active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers ■ HEROS.PrinterAdmin ■ HEROS.ReadLogs ■ HEROS.SWUpdate ■ HEROS.SetNetwork ■ HEROS.SetTimezone ■ HEROS.UserAdmin	■ userbck ■ lpadmin ■ logread ■ swupdate ■ netadmin ■ tz ■ useradmin	■ 337 ■ 16 ■ 342 ■ 341 ■ 336 ■ 333 ■ 339
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

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

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	■ NC.ScheduleProgramRun	■ NCSchedulePgRun	■ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter. Exception: The **Legacy-User** does not have the NC.ApproveFsAxis right.		
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	■ NCEditNCPgmAdv ■ NCEditTableAdv	■ 327 ■ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	■ NC.RemoteProgramRun	■ NCRemotePgmRun	■ 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	■ NCConfigUserAdv ■ 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	■ NCDAServiceRead	■ 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 658

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

27.3 Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)

The control always outputs signals in the metric system with Process Tracking Interface (PTI), regardless of the unit of the NC program.

You can output the following signals with PTI:

Signal designator	Description
a act	Actual axis acceleration value [m/s²] or [°/s²], calculated via the position encoder. Actual spindle acceleration value [rpm/sec], calculated via the speed encoder.
a nom	Nominal axis acceleration value [m/s²] or [°/s²]. Nominal spindle acceleration value [rpm/sec]
v act	Actual axis feed rate [mm/min] or [°/min], calculated via the position encoder. Actual spindle speed value [rpm], calculated via the speed encoder.
v nom	Nominal value of the axis feed rate [mm/min] or [°/min]. Axis feed rate calculated from the difference between the nominal position values. The following error is not included
Feed rate F	Contouring feed rate [mm/min] or [°/min]
Block no.	Block number of the NC program for triggering
s act	Actual position [mm] or [°] with respect to the machine coordinate system M-CS
s nom	Nominal position according to the nominal position value filter [mm] or [°] with respect to the machine coordinate system M-CS
s diff	Following error of the position controller [mm] or [°]
PosDiff	Difference between position and speed encoder [mm] or [°]
j act	Actual jerk value [m/s³], for rotary axes in [°/s³]. Calculated via the position encoder.
j nom	Nominal jerk value [m/s³], for rotary axes in [°/s³].
GantryDiff	Difference between synchronous axes before and during coupling [mm]
Analog	Analog axis/spindle: Analog voltage = Nominal velocity value [V]
SyncPosDiff	Position difference during spindle synchronism (slave spindle) [°]
AFC Diag.	Diagnosis information for AFC (Adaptive Feed Control, option 45). The following parameters are available: ■ **Spindle utilization** ■ **Override factor** ■ **Cutting condition** ■ **Position** ■ **Status** ■ **Spindle util. accord. to CC** ■ **Load for breakage monitoring**

Signal designator	Description
	■ **Upper overload limit** ■ **Lower overload limit** ■ **Idle load of spindle** The sampling interval should match the control loop cycle
Thread quality	Thread groove deviates from programmed contour
actTcpWcsX	Current TCP position in X in the workpiece coordinate system
actTcpWcsY	Current TCP position in Y in the workpiece coordinate system
actTcpWcsZ	Current TCP position in Z in the workpiece coordinate system
actTOriwcsZ	Current tool orientation, Euler angle Z
actTOriWcsX'	Current tool orientation, Euler angle X'
actTOriWcsZ''	Current tool orientation, Euler angle Z''
PLCPrePgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded **before** the PLC program run. This means that the values of the operands are read at the beginning of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
PLCPostPgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded **after** the PLC program run. This means that the values of the operands are read at the end of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
Comp	Diagnosis information on non-linear and linear axis-error compensations as well as the effective temperature compensation [mm or °]
kinemComp	Compensation value of KinematicsComp and the kinematic temperature compensation [mm]
tempComp	Pending compensation value of the lag-tracked PLC temperature compensation (PLC Module 9231) [mm or °].
backlashComp	Active compensation of reversal error [mm]
v act vctrl	Actual speed value on the output side, calculated from the motor encoder. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
v nom vctrl	Nominal speed value on the output side at position controller output. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
Iq int	Integral-action component of nominal current value [A]; CC 61xx/CC 3xx: effective value
Iq nom	Torque-defining nominal current value [A]; CC 61xx/CC 3xx: effective value
I2t mot	Current value of the I^2t monitoring of the motor [%]
I2t inv	Current value of the I^2t monitoring of the power module [%]
util-mot	Utilization of a motor with respect to the nominal point [%]

Signal designator	Description
util-DriveTrain	Utilization of the drive train in percent (power module + motor) [%]
J total	Estimated mass moment of inertia [kg * m²] (for rotary axes) from Load Adaptive Control (LAC—software option 143), if active
m total	Estimated mass [kg] (for linear axes) from Load Adaptive Control (LAC), if active
I act	Momentary actual current of the motor [A]
Id act	Momentary actual magnetizing current of the motor [A]
P mot elec	Electric power consumption of the motor [W]
P UV	Actual power released by the supply module [W]
U nom	Nominal voltage [V]
P mot mech	Mechanical power [W]
M act	Actual torque value [Nm]
P ZK	DC-link power of the power supply module [W]
U DC-link	DC-link voltage [V]
n act	Momentary motor speed [rpm]
n nom	Nominal value of motor speed [rpm]
Brake	Brake is released
s diff Ctrl	Servo lag of the position controller [mm] or [°]
U DC-link (UVR)	UVR DC-link voltage [V]
I DC-link	DC-link current [A]
I L1	Current of phase conductor L1 [A]
I L2	Current of phase conductor L2 [A]
I L3	Current of phase conductor L3 [A]
I leak	Leakage current [A]
P in	Supply unit power consumption [W]
U AC	Emulated line voltage [V]
f AC	Emulated mains frequency [Hz]
I out 24V	Current of the 24 V supply bar

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
1358774-24 · Ver04 · 10/2025 · H · Printed in Germany