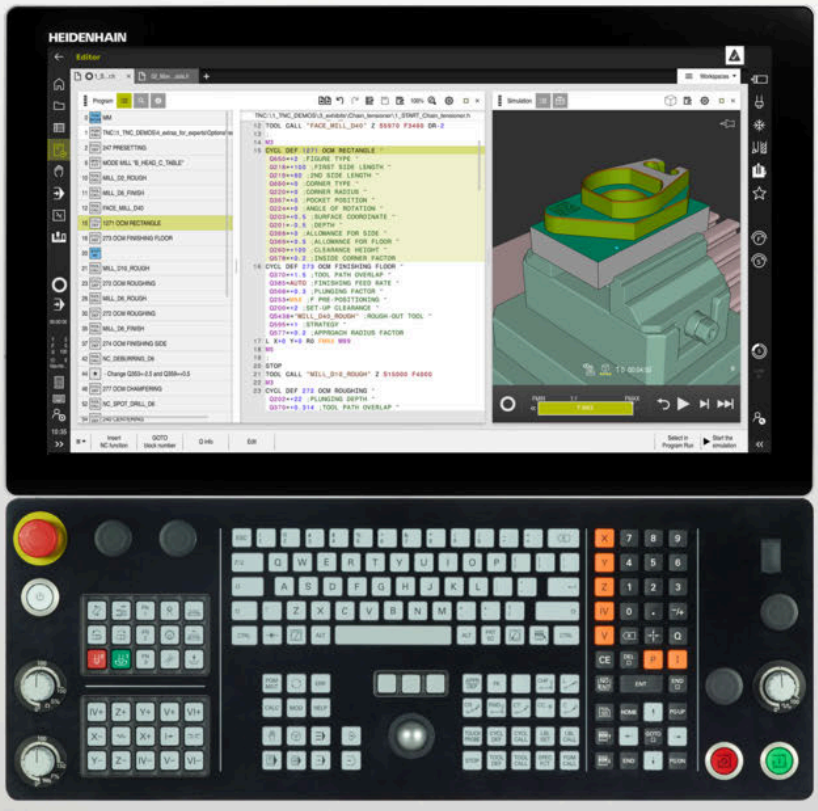
HEIDENHAIN



TNC7

User's Manual
Setup and Program Run

NC Software
81762x-17

English (en)
10/2022

Contents

1

New and Modified Functions

New functions 81762x-17

Further information: User's Manual for Programming and Testing

- You can run and edit ISO programs.
- In the text editor mode, the control provides an auto-complete function when programming. The control suggests syntax elements matching your entries which you can apply to the NC program.
- If an NC block contains a syntax error, the control displays a symbol in front of the block number. When you select the symbol, the control displays the corresponding error description.
- In the **Klartext** area of the **Program settings** window, you select whether the control skips the offered optional syntax elements of an NC block during input.

  If the toggle switches in the **Klartext** area are active, the control skips the syntax elements Comment, Tool index and Linear superimposition.
- If the control does not process or simulate the miscellaneous function **M1** or NC blocks hidden with **/**, it then grays out the miscellaneous function or NC blocks.
- When programming circular paths with **C**, **CR** and **CT**, the **LIN_** syntax element is now available in order to superimpose a linear motion over the circular motion of an axis. This allows you to program a helix in a simple way.

  In ISO programs, you can define a third axis in conjunction with the **G02**, **G03**, and **G05** functions.
- You can save up to 200 successive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. In contrast to the called NC programs, you can adapt the NC sequences after insertion without changing the actual sequence.
- The **FN 18: SYSREAD** (ISO: **D18**) functions have been enhanced:
  - **FN 18: SYSREAD (D18) ID610 NR49**: Mode of filter reduction of one axis (**IDX**) for **M120**
  - **FN 18: SYSREAD (D18) ID780**: Information on the current grinding tool
    - **NR60**: Active compensation method in **COR_TYPE** column
    - **NR61**: Inclination angle of dressing tool
  - **FN 18: SYSREAD (D18) ID950 NR48**: Value in column **R_TIP** in the tool table for the current tool
  - **FN 18: SYSREAD (D18) ID11031 NR101**: File name of the log file of Cycle **238 MEASURE MACHINE STATUS**
- In the **Visualization options** column of the **Simulation** workspace, you can show the worktable and, if necessary, the fixtures, in **Workpiece** mode and with the **Clamping situation** toggle switch.
- In the context menu of the **Editor** operating mode and the **MDI** application, the control offers the **Insert last NC block** function. With this function you can insert the last deleted or edited NC block in any NC program.

- You can perform file functions in the **Save as** window using the context menu.
- When you add a favorite or lock a file in the file management, the control displays an icon next to the file or folder.
- The **Document** workspace has been added. In the **Document** workspace, you can open files in order to view them, such as a technical drawing.
- Software option 159 (Model Aided Setup) has been added.

  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 clamping situation and possible touch points in the **Simulation** workspace by means of a 3D model.

  Further information: "Setting up the workpiece with graphical support (option 159)", Page 356
- If you execute an NC program or a pallet table or if you test it in the opened **Simulation** workspace, the control displays a navigation path in the file information bar of the **Program** workspace. The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

  Further information: "Navigation path in the Program workspace", Page 376
- The **TRANS** tab of the **Status** workspace indicates the active shift in the working plane coordinate system **WPL-CS**. If the shift comes from a compensation table (***.WCO**), the control shows the path to the compensation table as well as the number and, if applicable, the comment of the active row.

  Further information: "TRANS tab", Page 126
- You can transfer tables from earlier control models to the TNC7. If columns are missing in the table, the control opens the **Incomplete table layout** window.

  Further information: "Tables operating mode", Page 394
- The **Form** workspace in the **Tables** operating mode has been expanded:
  - The control displays an icon of the selected tool type in the **Tool Icon** area. For the turning tools the icons also take into account the tool orientation and show where the relevant tool data will apply.
  - Use the up and down arrows in the title bar to select the previous or next table row.

  Further information: "Form workspace for tables", Page 403
- You can create user-defined filters for the tool tables and pocket table. To do this, define a search condition in the **Search** column which you save as a filter.

  Further information: "Search column in the Table workspace", Page 400

- The following tool types have been added:
  - **Face mill** (**MILL_FACE**)
  - **Chamfer cutter** (**MILL_CHAMFER**)

  Further information: "Tool types", Page 168
- You define a database ID for the tool in the DB_ID column of the tool table. In a tool database for all machines, you can identify tools with unique database IDs (e.g., within a workshop). This allows you to coordinate the tools of multiple machines more easily.

  Further information: "Database ID", Page 164
- You define a radius at the tip of the tool in the **R_TIP** column of the tool table.

  Further information: "Tool table tool.t", Page 405
- You define the shape of the stylus in the **STYLUS** column of the touch probe table. You define an L-shaped stylus with the **L-TYPE** selection.

  Further information: "Touch probe table tchprobe.tp", Page 431
- You define the compensation method for dressing in the **COR_TYPE** input parameter for grinding tools (option 156):
  - **Grinding wheel with compensation, COR_TYPE_GRINDTOOL**

    Stock removal on the grinding tool
  - **Dressing tool with wear, COR_TYPE_DRESSTOOL**

    Stock removal on dressing tool

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 419
- Each user can create and activate configurations in which the control's user interface is individually adapted.

  You can save and activate individual modifications to the control's user interface as a configuration, e.g. for each operator. The configuration contains, for example, favorites and the arrangement of the workspaces.

  Further information: "Configuring the control's user interface", Page 547
- The **OPC UA NC Server** enables client applications to access the tool data of the control. You can read and write tool data.

  The **OPC UA NC Server** does not provide access to the grinding and dressing tool tables (option 156).

  Further information: "OPC UA NC Server (options 56 to 61)", Page 513
- Use the machine parameter **stdTNChelp** (no. 105405) to define whether the control displays help graphics as pop-up windows in the **Program** workspace.
- The optional machine parameter **CfgGlobalSettings** (no. 128700) allows you to define whether the control offers the parallel axes for **Handwheel superimp.**.

  Further information: "Function Handwheel superimp.", Page 269

New cycle functions with 81762x-17

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

- Cycle **1416 INTERSECTION PROBING** (ISO: **G1416**)

  This cycle allows you to determine the intersection of two edges. The cycle requires a total of four touch points and two positions per edge. You can use the cycle in the three object planes **XY**, **XZ** and **YZ**.
- Cycle **1404 PROBE SLOT/RIDGE** (ISO: **G1404**)

  This cycle determines the center and the width of a slot or ridge. The control probes two opposing touch points. You can also define a rotation for the slot or the ridge.
- Cycle **1430 PROBE POSITION OF UNDERCUT** (ISO: **G1430**)

  This cycle determines a single position with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus.
- Cycle **1434 PROBE SLOT/RIDGE UNDERCUT** (ISO: **G1434**)

  This cycle determines the center and the width of a slot or ridge with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes two opposing touch points.

Changed functions 81762x-17

Further information: User's Manual for Programming and Testing

- If you press the **actual position capture** key in the **Editor** operating mode or the **MDI** application, the control creates a straight line **L** with the current position of all axes.
- If you select the tool with the selection window when calling the tool with **TOOL CALL**, you can switch via an icon to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.
- You can use the **TABDATA** functions for read- and write-access to the preset table.
- If you define a grinding tool (option 156) with orientation **9** or **10**, the control supports circumferential milling in conjunction with **FUNCTION PROG PATH IS CONTOUR** (option 9).
- When you save an input value, the control removes superfluous zeros at the beginning of the input and at the end of the decimal places. The input range must not be exceeded for this.
- The control no longer interprets tab characters as syntax errors. In comments and structure items, the control displays a tab character as a space. In syntax elements, the control removes a tab character.
- If you edit a value and press the backspace key, the control deletes only the last character and not the complete input.
- You can delete an empty line with the backspace key in text editor mode.
- The **Insert NC function** window has been expanded:
  - In the areas **Search result**, **Favorites** and **Last functions**, the control shows the path of the NC functions.
  - If you select an NC function and swipe to the right, the control displays the following file functions:
    - Add to or remove from favorites
    - Open containing folder

    Only when you search for an NC function
  - If software options are not enabled, the control shows unavailable contents in the **Insert NC function** window grayed out.
- Graphical programming has been enhanced:
  - If you select the face of a closed contour, you can insert a radius or chamfer at each corner of the contour.
  - In the Element Information area, the control shows a rounding arc as **RND** contour element and a chamfer as **CHF** contour element.

- For a screen output with **FN 16: F-PRINT** (ISO: **D16**), the control displays a pop-up window.
- The window **Q parameter list** contains an input field that allows you to navigate to a unique variable number. If you press the **GOTO** key, the control selects the input field.
- The structure of the **Program** workspace has been enhanced:
  - The structure contains the NC functions **APPR** and **DEP** as structure elements.
  - The control shows comments in the structure inserted within structure elements.
  - If you mark structuring items in the **Structure** column, the control propagates the marking to the corresponding NC blocks in the NC program. Use the **CTRL+SPACE** key shortcut to stop marking. If you press **CTRL+SPACE** again, the control restores the marked selection.
- The **Search** column in the **Program** workspace has been enhanced:
  - The **Match whole words only** checkbox determines that the control shows only exact matches. If, for example, you search for **Z+10**, the control ignores **Z +100**.
  - If in the **Search and replace** function you use **Find next**, the control highlights the first result in purple.
  - If you do not enter a value for **Replace with:**, the control deletes the value searched for and to be replaced.
- If you select several NC blocks during the program comparison, you can load all NC blocks simultaneously.
- The control provides additional keyboard shortcuts to mark NC blocks and files.
- When you open or save a file in a selection window, the control displays the context menu.
- The cutting data calculator has been enhanced:
  - You can load the tool name from the cutting data calculator.
  - If you press the enter key in the cutting data calculator, the control selects the next element.

- The **Workpiece position** window of the **Simulation** workspace has been enhanced:
  - You can use a button to select a workpiece preset from the preset table.
  - The control displays the input fields below each other instead of next to each other.
- The control can display a finished part in the **Machine** mode of the **Simulation** workspace.
- The control takes into account the following columns of the tool table for the simulation:
  - **R_TIP**
  - **LU**
  - **RN**
- In the Simulation function of the **Editor** operating mode, the control takes dwell times into account. The control does not dwell during the program test, but adds the dwell times to the program run time.
- The NC functions **FUNCTION FILE** and **FN 27: TABWRITE** (ISO: **D27**) are active in the **Simulation** workspace.
- File management has been enhanced:
  - The control shows the occupied memory and total memory of the drives in the navigation bar of the file management.
  - The control shows STEP files in the preview area.
  - When you cut a file or folder in the file management, the control grays out the icon of the file or folder.
- The **Quick selection** workspace has been enhanced:
  - Tables for execution and simulation can be opened in the **Quick selection** workspace in the **Tables** operating mode.
  - In the **Quick selection** workspace in the **Editor** operating mode, you can create NC programs with mm or inch units of measurement as well as ISO programs.
- If you check the pallet table in Batch Process Manager (option 154) with Dynamic Collision Monitoring (DCM, option 40), the control takes the software limit switches into account.
- If you shut down the control with still unsaved changes in NC programs and contours, the control displays the **Close the program** window. You can save the changes, discard them or cancel the shutdown.

  Further information: "Powering off", Page 142
- You can change the size of windows. The control remembers the size until it is shut down.

  Further information: "Icons on the control's user interface", Page 91

- In the **Files**, **Tables** and **Editor** operating modes, a maximum of ten tabs can be open at the same time. If you try to open additional tabs, the control shows a message.

  Further information: "Areas of the control's user interface", Page 78
- **CAD-Viewer** has been enhanced:
  - Internally, **CAD-Viewer** always uses mm for its calculations. If you select inches as the unit of measure, **CAD-Viewer** will convert all values to inches.
  - The **Show sidebar** icon enlarges the Sidebar window to half the size of the screen.
  - The control always shows the **X**, **Y** and **Z** coordinates in the Element Information window. In 2D mode, the control grays out the Z coordinate.
  - **CAD-Viewer** also recognizes circles that consist of two semi-circles as machining positions.
  - You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

  Further information: "Opening CAD Files with the CAD-Viewer", Page 301
- The **Open in the editor** button in the **Program Run** operating mode opens the currently displayed NC program, including called NC programs.

  Further information: "Program Run operating mode", Page 368
- In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

  Further information: "Manual traverse during an interruption", Page 378
- Process monitoring (option 168) has been enhanced:
  - The **Process Monitoring** workspace contains a setup mode. When the mode is inactive, the control hides all functions for setting up process monitoring.

    Further information: "Icons", Page 277
  - When the settings of a monitoring task are selected, the control shows two areas with the original settings and the current settings of the monitoring task.

    Further information: "Monitoring tasks", Page 282
  - The control shows the coverage, i.e. the matching of the current graph with the graph of the reference machining, as circular charts.

    The control shows reactions of the notification menu in the graph and in the table of recordings.

    Further information: "Records of monitoring sections", Page 294

- The status overview on the TNC bar has been enhanced:
  - In the status overview, the control displays the run time of the NC program in mm:ss format. As soon as the run time of the NC program exceeds 59:59, the control shows the run time in hh:mm format.
  - If a tool usage file is available, the control calculates for the **Program Run** operating mode how long the execution of the active NC program will take. During program run the control updates the remaining run time. The control shows the remaining run time in the status overview on the TNC bar.
  - If more than eight axes are defined, the control shows the axes in two columns in the position display of the status overview. With more than 16 axes, the control shows the axes in three columns.

  Further information: "Status overview on the TNC bar", Page 115
- The control shows a feed rate limit in the status display:
  - If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the control shows the feed rate in orange.
  - If the feed rate is limited using the **FMAX** button, the control displays **MAX** in square brackets.

    Further information: "Feed rate limit FMAX", Page 372
  - If the feed rate is limited using the **F limited** button, the control displays the active safety function in square brackets.

    Further information: "Safety functions", Page 482
- In the **Tool** tab of the **Status** workspace, the control displays the values of the **Tool geometry** and **Tool allowances** areas with four instead of three decimal places.

  Further information: "Tool tab", Page 129
- If a handwheel is active, the control shows the contouring feed rate in the display during program run. If only the currently selected axis is moving, the control shows the axis feed rate.

  Further information: "Contents of an electronic handwheel display", Page 462

- If you align the rotary table after a manual touch probe function, the control remembers the selected type of rotary axis positioning and the feed rate.

  Further information: "Buttons", Page 336
- If you correct the preset or datum after a manual touch probe function, the control shows a symbol behind the adopted value.

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 331
- In the **3-D rotation** window (option 8), if you enable a function in the **Manual Operation** or **Program run** areas, the control highlights the area in green.

  Further information: "3-D rotation window (option 8)", Page 220
- The **Tables** operating mode has been enhanced:
  - The **M** and **S** statuses are highlighted in color only for the active application, and gray for the other applications.
  - You can close all applications except for **Tool management**.
  - The **Mark row** button has been added.
  - In the **Presets** application, the **Lock record** toggle switch has been added.

  Further information: "Tables operating mode", Page 394
- The **Table** workspace has been enhanced:
  - You can change the column width using an icon.
  - In the settings of the **Table** workspace you can enable or disable all table columns and restore the default format.

  Further information: "Table workspace", Page 396
- If a table column offers two input options, the control shows the options in the **Form** workspace as toggle switches.
- The minimum input value of the **FMAX** column in the touch probe table has been changed from –9999 to +10.

  Further information: "Touch probe table tchprobe.tp", Page 431
- You can import tool tables of the TNC 640 as CSV files.

  Further information: "Importing tool data", Page 186

- The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased. It was from 0 to 0.9999 mm, and is now from 0.0000 to 5.0000 mm.
- The maximum input range of the **LBREAK** and **RBREAK** columns of the tool table has been increased. It was from 0 to 3.2767 mm, and is now from 0.0000 to 9.0000 mm.

  Further information: "Tool table tool.t", Page 405
- If you double tap or click a tool in the **Tool check** column of the **Program** workspace, the control switches to the **Tables** operating mode. In this case, the control displays the selected tool in the **Tool management** application.

  Further information: "Tool check column in the Program workspace", Page 192
- In the expanded notification menu, the control displays information about the NC program in a separate area outside of the **Details**.

  Further information: "Message menu on the information bar", Page 328
- You can use the **Update documentation** function to install or update, for example, the **TNCguide** integrated product aid.

  Further information: "Update documentation", Page 540
- The control no longer supports the ITC 750 additional operating station.
- When you enter a code number in the **Settings** application, the control displays a load icon.

  Further information: "Code numbers", Page 493
- In the **DNC** menu item of the **Settings** application, the **Secure connections for users** area has been added. These functions can be used to define settings for secure connections via SSH.

  Further information: "Secure connections for user", Page 519
- 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 needing to transmit them to the control.

  Further information: "SSH-secured DNC connection", Page 568
- You can export and import existing network configurations in the **Network settings** window.

  Further information: "Exporting and importing a network profile", Page 512
- The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

  When the control detects a non-secure connection, it displays an informational notice.
- The optional machine parameter **warningAtDEL** (no. 105407) is used to define whether the control shows a confirmation request in a pop-up window when deleting an NC block.

Modified cycle functions with 81762x-17

Further information: User's Manual for Machining Cycles

- You can edit and execute Cycle **19 WORKING PLANE** (ISO: **G80**, option 8), but you cannot insert it into an NC program as a new element.
- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167) monitors contour damage on the floor caused by the tool tip. This tool tip results from the radius **R**, the radius at the tool tip **R_TIP**, and the point angle **T-ANGLE**.
- The parameter **Q592 TYPE OF DIMENSION** has been added to Cycle **292 CONTOUR.TURNG.INTRP.** (ISO: **G292**, option 96). This parameter is used to define whether the contour is programmed with radius dimensions or diameter dimensions.
- The following cycles consider the miscellaneous functions **M109** and **M110**:
  - Cycle **22 ROUGH-OUT** (ISO: G122)
  - Cycle **23 FLOOR FINISHING** (ISO: G123)
  - Cycle **24 SIDE FINISHING** (ISO: G124)
  - Cycle **25 CONTOUR TRAIN** (ISO: G125)
  - Cycle **275 TROCHOIDAL SLOT** (ISO: G275)
  - Cycle **276 THREE-D CONT. TRAIN** (ISO: G276)
  - Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167)
  - Cycle **277 OCM CHAMFERING** (ISO: G277, option 167)
  - Cycle **1025 GRINDING CONTOUR** (ISO: G1025, option 156)

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

- If KinematicsComp (software option 52) is active, the log of Cycle **451 MEASURE KINEMATICS** (ISO: **G451**, option 48) shows the active compensations of the angular position errors (**locErrA**/**locErrB**/**locErrC**).
- The log of Cycles **451 MEASURE KINEMATICS** (ISO: **G451**) and **452 PRESET COMPENSATION** (ISO: **G452**, option 48) contains diagrams with the measured and optimized errors of the individual measuring positions.
- Cycle **453 KINEMATICS GRID** (ISO: **G453**, option 48) allows you to use the mode **Q406=0** even without KinematicsComp (software option 52).
- Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) determines the radius and, if required, the length, the center offset and the spindle angle of an L-shaped stylus.
- Cycles **444 PROBING IN 3-D** (ISO: **G444**) and **14xx** support probing with an L-shaped stylus.

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. ability to read technical drawings and spatial imagination
- Basic knowledge in the field of metal cutting, e.g. meaning of material-specific parameters
- Safety instructions, e.g. possible dangers and their avoidance
- Training on the machine, e.g. axis directions and 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 User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator's manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a printed version, sub-divided into the modules below:
  - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for running NC programs.
    ID: 1358774-xx
  - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe and machining cycles are not included.
    ID for Klartext programming: 1358773-xx
  - The **Machining Cycles** User's Manual contains all functions of the machining cycles.
    ID: 1358775-xx
  - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
    ID: 1358777-xx
- As PDF files, sub-divided according to the printed versions or as a **Complete edition** User's Manual, containing all modules
  ID: 1369999-xx
  TNCguide
- As an HTML file used as the **TNCguide** product aid integrated directly into the control.
  TNCguide

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended use", Page 59

Further information products for users

The following information products are available to you:

- **Overview of new and modified software functions** informs you about the innovations of specific software versions.
  TNCguide
- **HEIDENHAIN brochures** inform you about products and services by HEIDENHAIN, e.g. software options of the control.
  HEIDENHAIN brochures
- The **NC solutions** database offers solutions for frequently occurring tasks.
  HEIDENHAIN NC solutions

2.3 Types of notes used

Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠ DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠ WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠ CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements contain 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 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, e.g. the documentation of your machine manufacturer or other supplier.

2.4 Notes on using NC programs

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

Change the following contents as needed:

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

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

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

	With a program test you determine whether the NC program can be used with the available software options, the active machine kinematics and the current machine configuration.

2.5 User's Manual as integrated product aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 49

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended use", Page 59

Requirement

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

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

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

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

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

Description of function

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

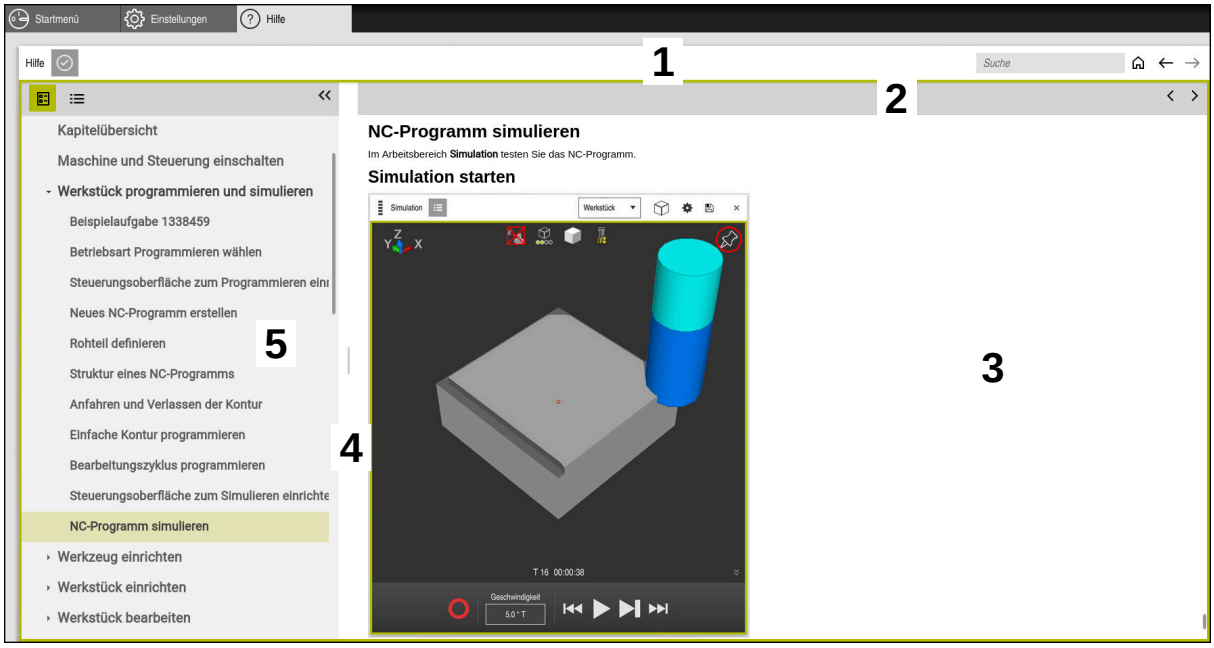
Further information: "Help application", Page 53

Further information: User's Manual for Programming and Testing

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

Further information: "Symbols", Page 54

Help application



Help application with open **TNCguide**

The **Help** application includes the following areas:

1 Title bar in the **Help** application
Further information: "Symbols in the Help application", Page 54

2 Title bar of the integrated product aid **TNCguide**
Further information: "Symbols in the integrated product aid TNCguide", Page 54

3 Content column of **TNCguide**

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

5 Navigation column of **TNCguide**

Symbols

Symbols in the Help application

Symbol	Function
	Show start 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.
	Display tutorials
 	Navigate between the last opened contents
	Display or hide search results **Further information:** "Search in TNCguide", Page 55

Symbols in the integrated product aid TNCguide

Symbol	Function
	Display documentation structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Display documentation index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
 	Display previous or next page within the documentation
 	Display or hide the navigation
	Copy NC examples to clipboard **Further information:** "Copying NC examples to clipboard", Page 55

2.5.1 Search in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

- Enter a character string

> The entry field is located in the title bar, to the left of the Home symbol that you use for navigating to the start page.
>
> The search starts automatically after you have entered e.g. a letter.
>
> 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 51

- 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: User's Manual for Programming and Testing

2.6 Contacting the editorial staff

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

We continuously strive to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

3

About the Product

3.1 The TNC7

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

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

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

Further information: User's Manual for Programming and Testing

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

Further information: "Hardware", Page 71

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 and the control, read the OEM documentation in order to inform yourself about the safety-relevent factors, the necessary safety equipment and the requirements for qualified personnel.

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

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

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

Further information: "Contacting the editorial staff", Page 55

3.1.2 Intended place of operation

In accordance with the DIN EN 50370-1 standard for electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

3.2 Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer's documentation!

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.

Refer to your machine manual.

Before using the machine and the control, read the OEM documentation in order to inform yourself about the safety-relevent factors, the necessary safety equipment and the requirements for qualified personnel.

The following overview contains only the generally valid safety precautions. Comply with the additional safey precautions provided in the following chapters. Some of this information depends on the specific configuraton.

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

DANGER

Caution: hazard to the user!

The **AUTOSTART** function automatically starts the machining operation. Open machines with unsecured work envelopes pose a huge danger for the machine operator.

- Use the **AUTOSTART** function exclusively on enclosed machines

⚠ 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.

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

NOTICE

Danger of collision!

NC programs that were created on older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!

- Check the NC program or program section using the graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

NOTICE

Caution: Data may be lost!

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

- Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs
- Use the **Eject** soft key to remove a USB device when data the transfer is complete

NOTICE

Caution: Data may be lost!

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

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

NOTICE

Danger of collision!

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

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

3.3 Software

This User's Manual describes the functions for setting up the machine as well as for programming and and running your NC programs. These functions are available for a control featuring the full range of functions.

The actual range of functions depends, among other things, on the enabled software options.
Further information: "Software options", Page 64

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
817620-17	TNC7
817621-17	TNC7 E
817625-17	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 for the machine.

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

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, Advanced Function Set 2 (software option 9) is restricted 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 498

Overview and definitions

The **TNC7** features various software options, each of which can be enabled separately and even subsequently by the machine manufacturer. The following overview includes only those software options that are relevant for you as user.

> The option numbers indicated in the User's Manual show you that a function is not included in the standard range of available functions.
>
> The Technical Manual provides information about additional software options that are relevant for the machine manufacturer.

> Keep in mind that particular software options also require hardware extensions.
>
> **Further information:** "Hardware", Page 71

Software option	Definition and application
Additional Axis (options 0 to 7)	**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 e.g. for detachable and motor-driven tilting tables.
Advanced Function Set 1 (option 8)	**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:** User's Manual for Programming and Testing ■ Programming of contours on the unrolled surface of a cylinder (e.g., by using 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:** User's Manual for Programming and Testing ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.

Software option	Definition and application
Advanced Function Set 2 (option 9)	**Advanced functions (set 2)** On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces. This software option includes the following functions: ■ **TCPM** (tool center point management): Automatic tracking of linear axes during rotary axis positioning **Further information:** User's Manual for Programming and Testing ■ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** User's Manual for Programming and Testing ■ Manual moving of axes in the active tool coordinate system **T-CS** ■ Linear interpolation in more than 4 axes (max. 4 axes for an export version) The advanced functions (set 2) can be used to produce free-form surfaces.
HEIDENHAIN DNC (option 18)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data in the control via the TCP/IP protocol. Potential fields of application are e.g.: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Dynamic Collision Monitoring (option 40)	**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 the program run whenever a collision is imminent ■ Warnings in the event of manual axis movements ■ Collision monitoring in Test Run mode With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or a machine downtime. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 226
CAD Import (option 42)	**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:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312

Software option	Definition and application
Global PGM Settings (option 44)	**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:** "Globale Programmeinstellungen GPS", Page
Adaptive Feed Control (option 45)	**Adaptive Feed Control AFC** This software option enables 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 the machining time without adapting the NC program, while preventing machine damage from overload at the same time. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 252
KinematicsOpt (option 48)	**KinematicsOpt** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsOpt, the control can compensate for errors on rotary axes and thus increase accuracy during machining operations in the tilted working plane and during simultaneous machining operations. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools
Turning (option 50)	**Mill-turning** This software option offers a comprehensive turning-specific package of functions 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 radius compensation Mill-turning enables mill-turning machining operations on only one machine, thus reducing e.g. the setup work considerably. **Further information:** User's Manual for Programming and Testing
KinematicsComp (option 52)	**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 compensate for the errors of rotary and linear axes in three dimensions. Compared to KinematicsOpt (option 48), compensation is far more extensive. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools

Software option	Definition and application
OPC UA NC Server 1 to **6** (options 56 to 61)	**OPC UA NC Server** These software options offer the OPC UA standardized interface for external access to data and functions of the control. Potential fields of application are e.g.: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection each. Several parallel connections require the use of multiple OPC UA NC servers. **Further information:** "OPC UA NC Server (options 56 to 61)", Page 513
4 Additional Axes (option 77)	**4 additional control loops** **Further information:** "**Additional Axis** (options 0 to 7)", Page 64
8 Additional Axes (option 78)	**8 additional control loops** **Further information:** "**Additional Axis** (options 0 to 7)", Page 64
3D-ToolComp (option 92)	**3D-ToolComp** only in conjunction with Advanced Function Set 2 (option 9) With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated for using a compensation value table. 3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example. **Further information:** User's Manual for Programming and Testing
Extended Tool Management (option 93)	**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** to be run shows the tool requirements of the NC program to be run or the pallet shows the tool requirements of the NC program **Further information:** "Tooling list (option 93)", Page 442 ■ The **T usage order** shows the tool order of the NC program to be run or the pallet. **Further information:** "T usage order (option 93)", Page 440 Extended Tool Management enables you to detect the tool requirements in time and thus prevent interruptions during program run.
Advanced Spindle Interpolation (option 96)	**Interpolating Spindle** This software option enables interpolation turning by causing the control to couple the tool spindle with the linear axes. This software option includes the following cycles: ■ Cycle **291 COUPLG.TURNG.INTERP.** for simple turning operations without contour subprograms ■ Cycle **292 CONTOUR.TURNG.INTRP.** for finishing rotationally symmetrical contours The interpolating spindle also allows you to perform a turning operation on machines without a rotary table. **Further information:** User's Manual for Machining Cycles

Software option	Definition and application
Spindle Synchronism (option 131)	**Spindle synchronism** This software option synchronizes two or more spindles and thus enables e.g. gear manufacturing via hobbing. This software option includes the following functions: ■ Spindle synchronism for special machining operations, e.g. polygonal turning ■ Cycle **880 GEAR HOBBING** only in conjunction with mill-turning (option 50) **Further information:** User's Manual for Machining Cycles
Remote Desktop Manager (option 133)	**Remote Desktop Manager** This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase efficiency. **Further information:** "Remote Desktop Manager window (option 133)", Page 527
Dynamic Collision Monitoring v2 (option 140)	**Dynamic Collision Monitoring (DCM) version 2** This software option includes the functions of software option 40 (Dynamic Collision Monitoring, DCM). In addition, this software option can be used for the collision monitoring of workpiece fixtures. **Further information:** "Integrating the fixtures into collision monitoring (option 140)", Page 235
Cross Talk Compensation (option 141)	**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 Adaptive Control (option 142)	**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 Adaptive Control (option 143)	**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 Adaptive Control (option 144)	**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 Control (option 145)	**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 material-removal rate can be increased by more than 25 %. **Further information:** "Active Chatter Control (ACC, option 145)", Page 260

Software option	Definition and application
Machine Vibration Control (option 146)	**Vibration damping for machines MVC** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**
CAD Model Optimizer (option 152)	**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 (option 152)", Page 318
Batch Process Manager (option 154)	**Batch Process Manager BPM** This software option makes it easy to plan and execute multiple production jobs. If pallet management and extended tool management (option 93) are extended or combined, 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:** User's Manual for Programming and Testing
Component Monitoring (option 155)	**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 (option 156)	**Jig grinding** This software option offers a comprehensive grinding-specific package of functions for milling machines. The software option includes the following functions: ■ Grinding-specific tools including dressing tools ■ Cycles for reciprocating stroke and dressing Jig-turning enables complete machining operations on just one machine, thus reducing e.g. the setup work considerably. **Further information:** User's Manual for Programming and Testing
Gear Cutting (option 157)	**Gear manufacturing** This software option enables the manufacturing 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 (option 50). **Further information:** User's Manual for Machining Cycles

Software option	Definition and application
Turning v2 (option 158)	**Mill-turning version 2** This software option includes all functions of Mill-Turning (software option 50). 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 (option 159)	**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 clamping situation and possible touch points in the **Simulation** workspace by means of a 3D model.
Optimized Contour Milling (option 167)	**Optimized contour machining (OCM)** This software option enables the trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions. The software option includes the following cycles: ■ Cycle **271 OCM CONTOUR DATA** ■ Cycle **272 OCM ROUGHING** ■ Cycle **273 OCM FINISHING FLOOR** and cycle **274 OCM FINISHING SIDE** ■ Cycle **277 OCM CHAMFERING** ■ In addition, the control offers **OCM STANDARD FIGURES** for frequently needed contours With OCM you can shorten the machining time while reducing tool wear at the same time. **Further information:** User's Manual for Machining Cycles
Process Monitoring (option 168)	**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 with regard to the tool spindle or the tool with the values of a reference machining operation. **Further information:** "Arbeitsbereich Prozessüberwachung (Option #168)", Page

3.3.2 Information on licensing and use

Open-source software

The control software contains open-source software whose use is subject to explicit licensing terms. These special terms of use have priority.

To get to the licensing terms on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Operating system** tab
- Double-tap or double-click **About HeROS**
- The control opens the **HEROS Licence Viewer** window.

OPC UA

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

OPC UA NC Server (options 56 to 61) and HEIDENHAIN DNC (option 18) can be used to influence the behavior of the control. 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 (options 56 to 61)", Page 513

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 63

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

3.4.1 Monitor

BF 360

The TNC7 is delivered with a 24-inch touchscreen.

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

Further information: "Common gestures for the touchscreen", Page 85

Further information: "Operating elements of the keyboard unit", Page 85

Operation and cleaning

Avoiding electrostatic discharge when operating touchscreens

Touchscreens are based on a capacitive working principle, i.e. they are sensitive to electrostatic charges generated by the operators.

Users can discharge static electricity from their body by touching grounded metal objects. This problem can be avoided by wearing ESD clothing.

Capacitative sensors detect a contact as soon as a person's finger touches the touchscreen. Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. While small amounts of liquid will not cause a fault, larger quantities of liquid will cause erroneous input.

Use work gloves to prevent the device from becoming dirty. The rubber material of special touchscreen work gloves contains metal ions that transfer the skin resistance to the display.

In order to maintain the functionality of the touchscreen, use the following cleaners only:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

Do not apply the cleaner directly to screen, but slightly dampen a suitable cleaning cloth with it.

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

Further information: "Settings Application", Page 489

Never use the following cleaners or cleaning aids, in order to avoid damage to the touchscreen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

3.4.2 Keyboard unit

TE 360 with standard potentiometer layout

TE 360 with alternative potentiometer layout

TE 361

The TNC7 is delivered with various keyboard units.

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

Further information: "Common gestures for the touchscreen", Page 85

Further information: "Operating elements of the keyboard unit", Page 85

	Refer to your machine manual. Some machine manufacturers do not use the standard HEIDENHAIN operating panel. External keys, e.g. **NC START** or **NC STOP**, are described in your machine manual.

Cleaning

Use operating gloves to prevent the device from becoming dirty.

In order to maintain the functionality of the keyboard, use only cleaners stated to contain anionic or nonionic surfactants.

Do not apply the cleaner directly to the keyboard unit. Slightly dampen a suitable cleaning cloth with the cleaner.

Switch the control off before cleaning the keyboard unit.

Never use the following cleaners or cleaning aids, in order to avoid damage to the keyboard unit:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam blasters

The trackball does not require periodic maintenance. Cleaning is required only if the trackball stops functioning.

If a trackball is embedded in the keyboard, clean the trackball as follows:

- ▶ Switch off 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 an isopropyl alcohol cleaner to a lint-free and clean cloth

  Please observe the information for the cleaner.

- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

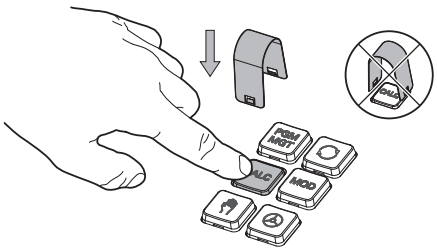
Exchanging keycaps

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

Further information: "Keycaps for keyboard units and machine operating panels", Page 608

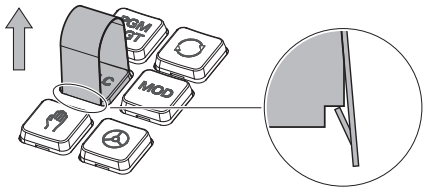
The IP54 protection rating cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

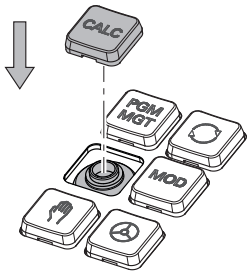


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

  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 the IP54 protection rating cannot be guaranteed.

- Verify proper seating and correct functioning

3.4.3 Hardware enhancements

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

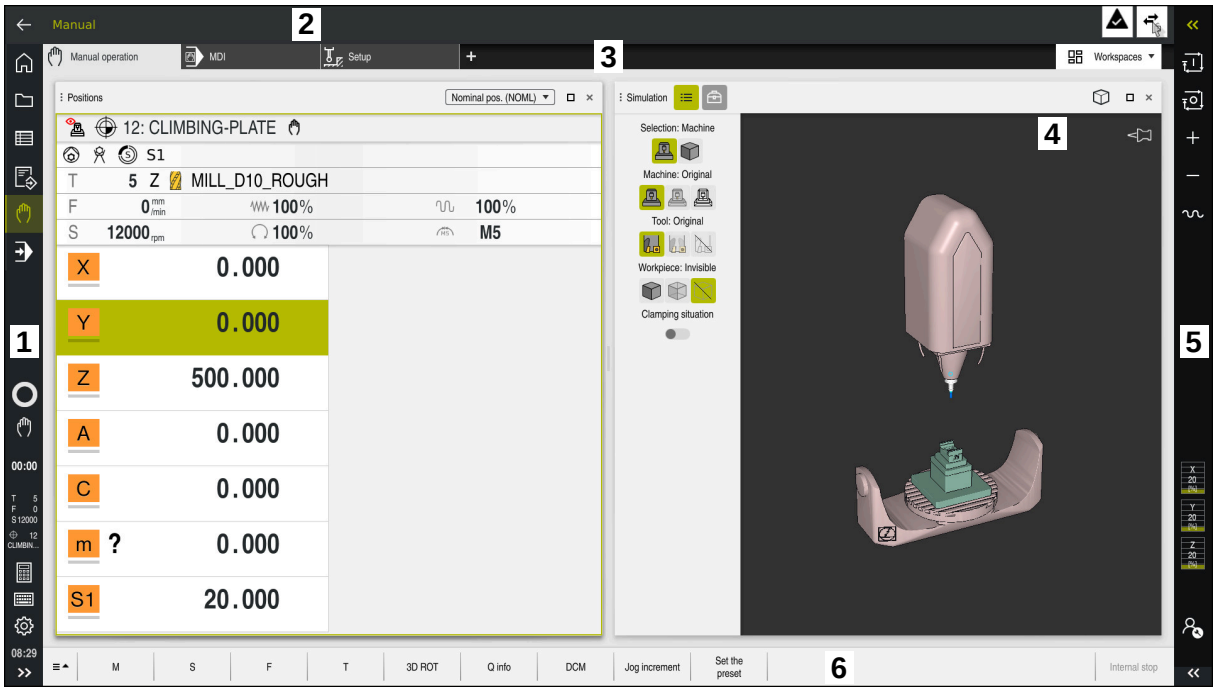
The **TNC7** features various hardware enhancements, each of which can be added separately and even subsequently by the machine manufacturer. The following overview includes only those enhancements that are relevant for you.

> Keep in mind that particular hardware enhancements require additional software options.
>
> **Further information:** "Software options", Page 64

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 459
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 331
Tool touch probes	The control uses this enhancement for automatic and precise calibration of tools 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:** User's Manual for Measuring Cycles for Workpieces and Tools

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 (option 133), you can display the applications on the control's screen. **Further information:** "Remote Desktop Manager window (option 133)", Page 527 The industrial PC is a secure and powerful alternative to external PCs.

3.5 Areas of the control's user interface



The control's user interface 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 operating modes", Page 79
- Status overview

  Further information: "Status overview on the TNC bar", Page 115
- Calculator

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

  Further information: "Virtual keyboard of the control bar", Page 324
- 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**
  - **Font size**
- Date and time

2 Information bar
- Active operating mode
- Message menu

  Further information: "Message menu on the information bar", Page 328
- Symbols

3 Application bar
- Tabs of opened applications

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

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

4 Workspaces

Further information: "Workspaces", Page 81

5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.

6 Function bar
- Selection menu for buttons

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

  With the buttons you activate individual functions of the control.

3.6 Overview of operating modes

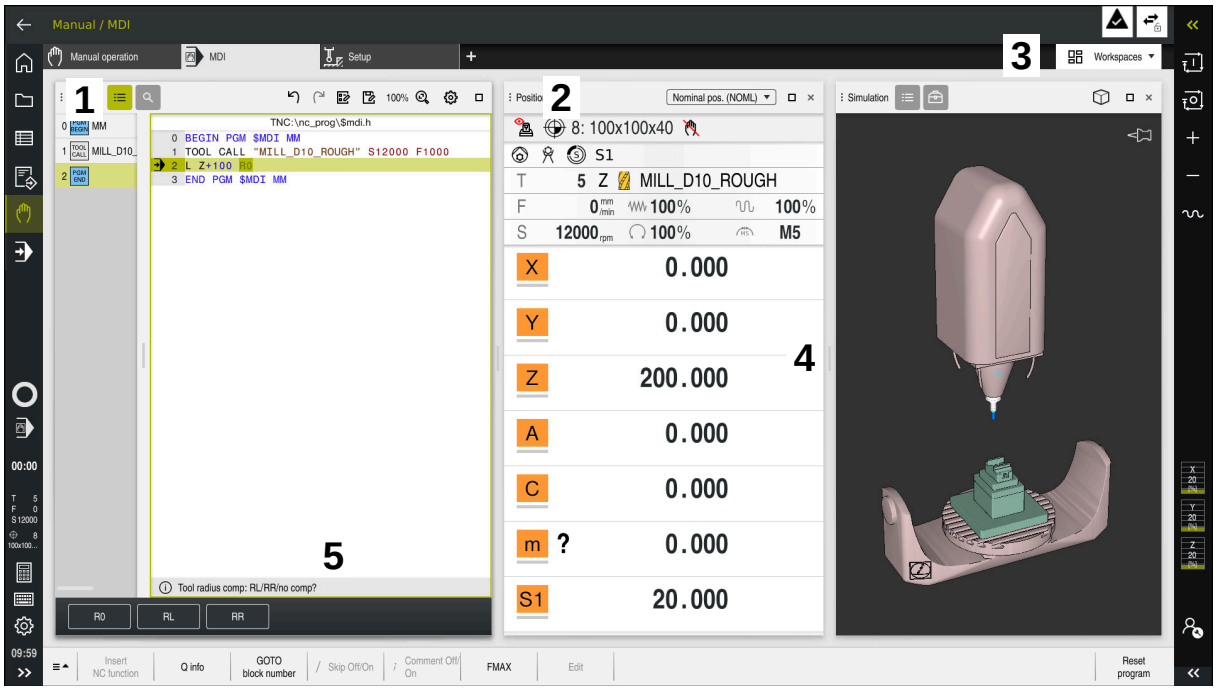
The control provides the following operating modes:

Symbols	Operating modes	Further information
	The **Home** operating mode contains the following applications: ■ **Start/Login** application During the startup process, the control is in the **Start/Login** application. ■ **Settings** application ■ **Help** application ■ Applications for machine parameters	 Page 489 See the User's Manual for Programming and Testing Page 542
	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 394
	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

Symbols	Operating modes	Further information
	The **Manual** operating mode contains the following applications:	
	■ **Manual operation** application	Page 146
	■ **MDI** application	Page 363
	■ **Setup** application	Page 331
	■ **Move to ref. point** application	Page 141
	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence. You also execute pallet tables in this operating mode.	Page 368
	In the **Retract** application you can move the tool away from the workpiece, for example after a power failure.	Page 390
	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 477
	In the **Machine** operating mode, the machine manufacturer defines his own functions, such as diagnostic functions for spindle and axes, or applications. Refer to your machine manual.	

3.7 Workspaces

3.7.1 Operating elements within the workspaces



The control in the **MDI** application with three open workspaces

The control displays the following operating elements:

1 Gripper

Use the gripper in the title bar to change positions of the workspaces. You can also align two workspaces vertically above each other.

2 Title bar

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

3 Selection menu for workspaces

Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.

4 Separator

You use the separator between two workspaces to change the scaling of the workspaces.

5 Action bar

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

3.7.2 Symbols within the workspaces

If more than one workspace is open, the title bar contains the following symbols:

Icon	Function
	Maximize workspace
	Reduce workspace
	Close workspace

If you maximize a workspace, the control shows the workspace over the application's entire area. If you reduce the workspace, then all other workspaces return to their previous position.

3.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function In the **Probing function** workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools, and set up fixtures.	Page 331
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 can select and create files, for example.	See the User's Manual for Programming and Testing
Document In the **Document** workspace, you can open files in order to view them, such as a technical drawing.	See the User's Manual for Programming and Testing
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 403
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 disengage the tool after a power interruption.	Page 390
GS (option 44) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	Page 261
Desktop menu In the **Desktop menu** workspace the control displays selected control and HEROS functions.	Page 93

Workspace	Further information
Help In the **Help** workspace the control displays a help graphic for the current syntax element of an NC function or the **TNCguide** integrated product aid.	See the User's Manual for Programming and Testing
Contour In the **Contour** workspace you use lines and arcs to draw a 2D sketch and then generate a Klartext contour from it. You can also import program sections with contours from an NC program to the **Contour** workspace for graphical editing.	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 543
Positions In the **Positions** workspace the control displays information about the status of various functions of the control and about current axis positions.	Page 109
Program The control displays the NC program in the **Program** workspace.	See the User's Manual for Programming and Testing
RDP (option 133) 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 477
Quick selection In the **Quick selection** workspace, you can create files or open existing ones regardless of the active operating mode.	See the User's Manual for Programming and Testing
Simulation In the **Simulation** workspace the control shows the simulated or actual 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 131
Start/Login In the **Start/Login** workspace the control shows the steps that are performed while booting.	Page 96
Status In the **Status** workspace the control shows the status and values of individual functions.	Page 117
Table In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.	Page 396
Table for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	Page 543

Workspace	Further information
Keyboard In the **Keyboard** workspace you can enter NC functions, letters and numbers, and also navigate.	Page 324
Overview In the **Overview** workspace the control displays the information on the status of individual functional safety (FS) aspects.	Page 484
Monitoring In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.	Page 276

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:

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of finger tip on the screen If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds. Permanent actuation is thus not possible.
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

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 85

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.

Keycaps for alphabetic keyboard

Key	Function
A B C	Enter texts (e.g., file names)
SHIFT + Q	**Uppercase Q** If an NC program is open, in the **Editor** operating mode for entering a Q parameter formula; in the **Manual** operating mode for opening the **Q parameter list** window **Further information:** User's Manual for Programming and Testing
ESC	Close windows and context menus
⇄	Select the next element, e.g. input field, button, selection option
SHIFT + ⇄	Select the previous element
PRT SC	Create screenshot
	**Left DIADUR key** Open the
	Open the context menu in **Klartext programming** or in the text editor

Keycaps for operating aids

Key	Function
PGM MGT	Open the **Open File** workspace in the **Editor** and **Program Run** operating modes **Further information:** User's Manual for Programming and Testing
	Select the first right-aligned button in the function bar
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 328
CALC	Open and close the calculator **Further information:** User's Manual for Programming and Testing
MOD	Open the **Settings** application **Further information:** "Settings Application", Page 489
HELP	Open the online help **Further information:** "User's Manual as integrated product aid: TNCguide", Page 52

Operating modes

On the TNC7 the operating modes of the control are allocated differently than on the TNC 640. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but, for example, instead activate a toggle switch.

Key	Function
	Opening the **Manual operation** application in the **Manual** operating mode **Further information:** "Manual operation application", Page 146
	Activating and deactivating the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic Handwheel", Page 459
	Opening the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 184
	Opening the **MDI** application in the **Manual** operating mode **Further information:** "The MDI Application ", Page 363
	Opening the **Program Run** operating mode in **Single Block** mode **Further information:** "Program Run operating mode", Page 368
	Opening the **Program Run** operating mode **Further information:** "Program Run operating mode", Page 368
	Opening the **Editor** operating mode **Further information:** User's Manual for Programming and Testing
	While the NC program is running, opening of the **Simulation** workspace in the **Editor** operating mode **Further information:** User's Manual for Programming and Testing

Keycaps for NC dialog

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

Key	Function
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode
CHF	Program a chamfer
L	Program a straight line segment
CR	Program a circular arc with radius entry
RND	Program a rounding arc
CT	Program a circular arc with tangential connection to the preceding contour element
CC	Program a circle center or pole
C	Program a circular arc with reference to the circle center
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools
CYCL DEF	In the **Insert NC function** window, open the **Fixed cycles** folder in order to select a cycle **Further information:** User's Manual for Machining Cycles
CYCL CALL	In the **Insert NC function** window, open the **Cycle call** folder in order to select a machining cycle **Further information:** User's Manual for Machining Cycles
LBL SET	Program a jump label
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 data in the NC program
SPEC FCT	In the **Insert NC function** window, open the **Special functions** folder (e.g., for later programming of a workpiece blank)
PGM CALL	In the **Insert NC function** window, open the **Selection** folder (e.g., to call an external NC program)

Further information: User's Manual for Programming and Testing

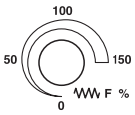
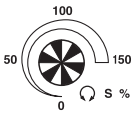
Keycaps for axis input and value input

Key	Function
X ... V	Select axes in the **Manual** operating mode, or enter them in the **Editor** operating mode
0 ... 9	Enter numbers (e.g., coordinate values)
.	Insert a decimal separator during entry
−/+	Invert algebraic sign of entered value
X	Delete values during entry
	Open position display of the status overview to copy axis values **Further information:** "Status overview on the TNC bar", Page 115 In **Editor** operating mode and the **MDI** application, program a straight line **L** with the actual positions of all axes
Q	In the **Editor** operating mode, within the **Insert NC function** window, open the **FN** folder
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	Function
	Position the cursor
	■ Position the cursor by using the block number of an NC block ■ 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

Poten-tiometer	Function
	Increase or reduce the feed rate **Further information:** User's Manual for Programming and Testing
	Increase or reduce the spindle speed **Further information:** User's Manual for Programming and Testing

3.8.3 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	Function
	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 and close the calculator
	Open and close the virtual keyboard
	Open and close the settings
	■ White: Expand control bar or machine manufacturer bar ■ Green: Collapse control bar or machine manufacturer bar or go back ■ Gray: Confirm message
	Add
	Open file
	Close
	Maximize workspace
	Reduce workspace
	Change position of workspaces or windows
	Resize windows
	■ Black: Add to favorites ■ Yellow: Remove from favorites

Icon or shortcut	Function
 CTRL+S	Save
	Save as
 CTRL+F	Find
 CTRL+C	Copy
 CTRL+V	Paste
 CTRL+Z	Undo an action
 CTRL+Y	Redo an action
	Open selection menu
	Open message menu

3.8.4 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:

- **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 547
- Full-text search

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

  Further information: "Adding and removing favorites", Page 94

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

- **Control**

  In this area you can open operating modes or applications.

  Further information: "Overview of operating modes", Page 79

  Further information: "Overview of workspaces", Page 82
- **Tools**

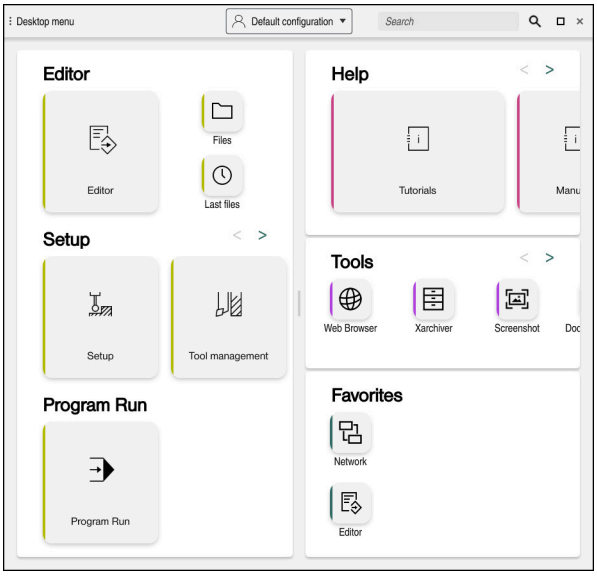
  In this area you can open some tools from the HEROS operating system.

  Further information: "HEROS Operating System", Page 573
- **Help**

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

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

  Further information: "Adding and removing favorites", Page 94



Desktop menu workspace

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

Showing or hiding an area

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

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

	Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

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

- ▶ Use the full-text search
- ▶ Hold or right-click the function's icon
- > The control displays the icon for **adding favorites**.

- ▶ Select **Add favorite**
- > The control adds the function to the **Favorites** area.

Removing favorites

To remove favorites from the **Desktop menu** workspace:

- ▶ Hold or right-click the function's icon
- > The control displays the icon for **removing favorites**.

- ▶ Select **Remove favorite**
- > The control removes the function from the **Favorites** area.

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 on
- Setting up tools
- Setting up the workpiece
- Machining the workpiece
- Switching the machine off

4.2 Switching the machine and the control on

Start/Login workspace

DANGER

Caution: hazard to the user!

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

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

Refer to your machine manual.

Switching on the machine and traversing the reference points can vary depending on the machine tool.

To switch the machine on:

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

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

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

  Further information: "Referencing workspace", Page 141

- ▶ 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: "Manual operation application", Page 146

More detailed information

- Switching on and off
- Position encoders

  Further information: "Position encoders and reference marks", Page 153
- Axis reference run

4.3 Configuring a tool

4.3.1 Selecting the Tables operating mode

You configure tools in the **Tables** operating mode.

To select the **Tables** operating mode:

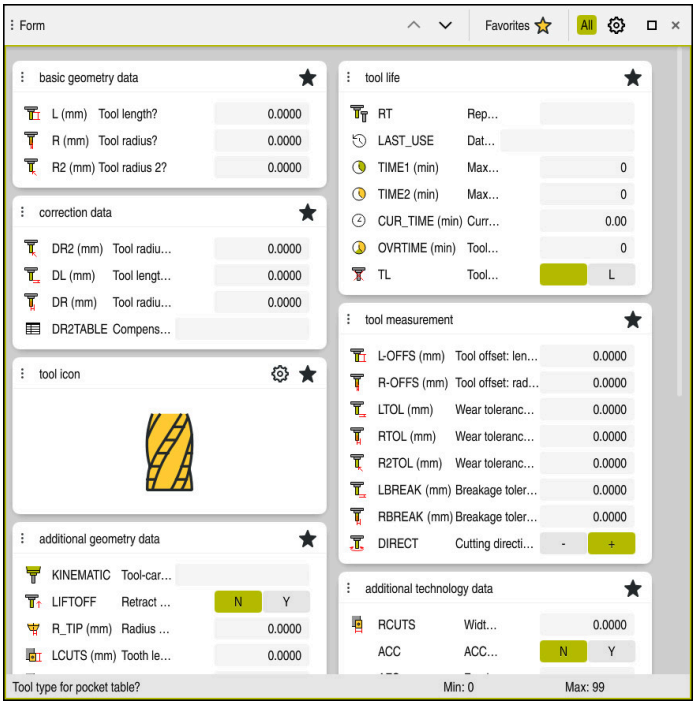
- ▶ Select the **Tables** operating mode
- > The control displays the **Tables** operating mode.

More detailed information

- Operating mode: **Tables**

  Further information: "Tables operating mode", Page 394

4.3.2 Configuring the control's user interface



Form workspace in the **Tables** operating mode

In the **Tables** operating mode you open and edit the various tables of the control either in the **Table** workspace or in the **Form** workspace.

	The first steps describe the procedure with the **Form** workspace open.

To open the **Form** workspace:

- In the application bar, select **Workspaces**
- Select **Form**

> The control opens the **Form** workspace.

More detailed information

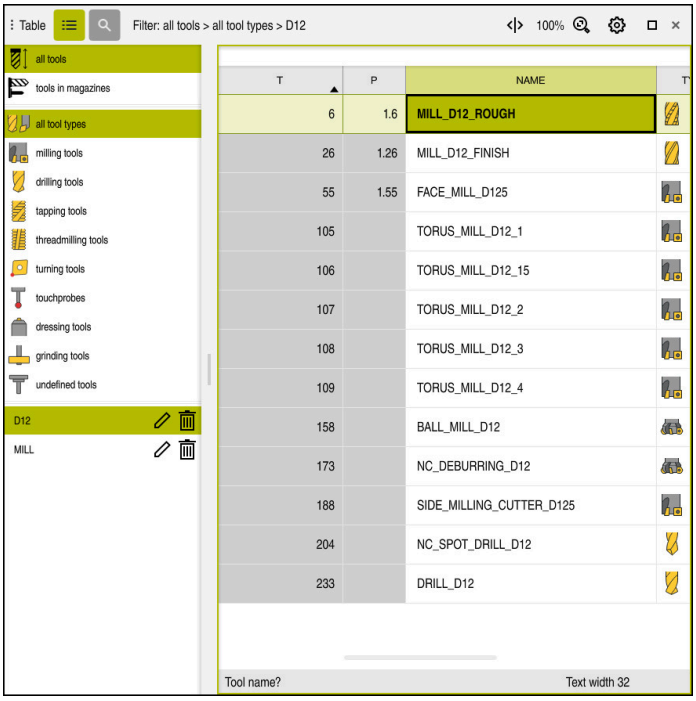
- **Form** workspace
  Further information: "Form workspace for tables", Page 403
- Workspace: **Table**
  Further information: "Table workspace", Page 396

4.3.3 Preparing and measuring tools

To prepare tools:

- Clamp the required tools in their tool holders
- Measure the tools
- Write down the length and the radius or transfer these directly to the control

4.3.4 Editing within tool management



Tool management application in the **Table** workspace

Tool management allows you to save tool data, such as the length and radius as well as other tool-specific information.

The control displays the tool data for all tool types in tool management. In the **Form** workspace the control displays only the relevant tool data for the current tool type.

To enter the tool data in tool management:

- ▶ Select **Tool management**
- > The control displays the **Tool management** application.
- ▶ Open the **Form** workspace

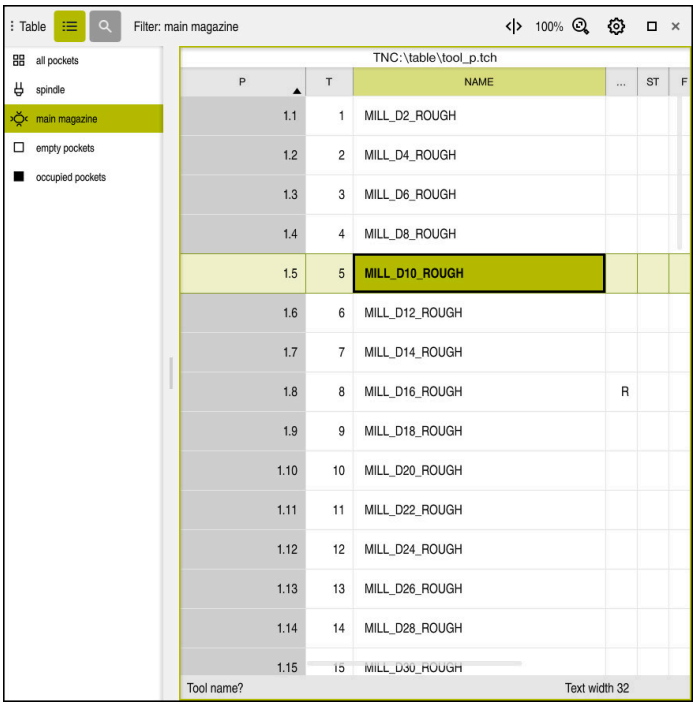
- ▶ Activate **Edit**
- ▶ Select the desired tool number (e.g., **16**)
- > The control displays the tool data of the selected tool in the form.
- ▶ Define the required tool data in the form; for example, the length **L** and the tool radius **R**

More detailed information

- Operating mode: **Tables**
  Further information: "Tables operating mode", Page 394
- Workspace: **Form**
  Further information: "Form workspace for tables", Page 403
- Tool management
  Further information: "Tool management ", Page 184
- Tool types
  Further information: "Tool types", Page 168

4.3.5 Editing the pocket table

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



Pocket table application in the **Table** workspace

The control assigns a pocket in the tool magazine to each tool that is in the tool table. This assignment, as well as the load situation of each tool, is shown in the pocket table.

There are various ways of accessing the pocket table:

- Functions of the machine manufacturer
- Third-party tool-management system
- Manual access to the control

To enter the data in the pocket table:

- ▶ Select **Pocket table**
- > The control displays the **Pocket table** application.
- ▶ Open the **Form** workspace

- ▶ Activate **Edit**
- ▶ Select the desired pocket number
- ▶ Define the tool number
- ▶ Define any additional tool data if necessary, such as whether the pocket is reserved

More detailed information

- Pocket table
  Further information: "Pocket table tool_p.tch", Page 435

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 operating modes", Page 79

4.4.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

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.
- Manually position the touch probe near the first touch point of the first workpiece edge

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

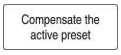
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the first workpiece edge

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

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

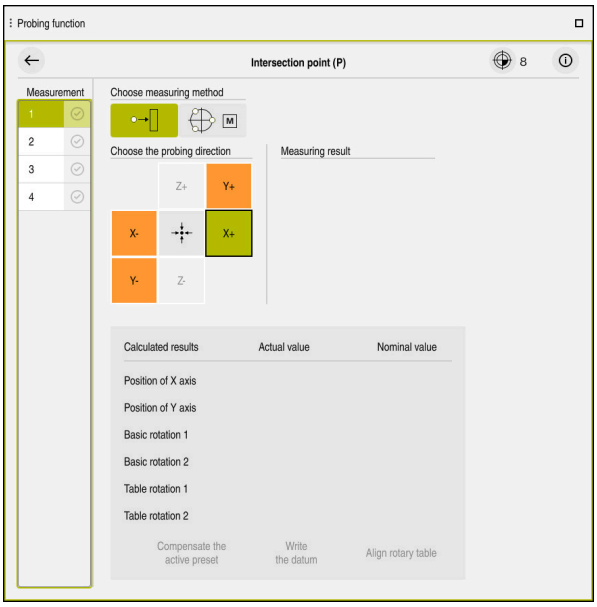
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the second workpiece edge

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



- Select **Compensate the active preset**
- The control applies the calculated results to the workpiece preset.
- The control highlights the line with a preset symbol.

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



Probing function workspace with an open manual probing function

More detailed information

- Workspace: **Probing function**

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 331
- Reference points in the machine

  Further information: "Presets in the machine", Page 154
- Tool change in the **Manual operation** application

  Further information: "Manual operation application", Page 146

4.5 Machining a workpiece

4.5.1 Selecting an operating mode

You 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

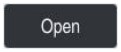
- Operating mode: **Program Run**
  Further information: "Program Run operating mode", Page 368

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

- Workspace: **Open File**
  Further information: User's Manual for Programming and Testing

4.5.3 Starting an NC program

To start an NC program:

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

4.6 Switching the machine off

Refer to your machine manual.
Switching off is a machine-dependent function.

NOTICE

Caution: Data may be lost!

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

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

To power-off the machine:

- Select the **Home** operating mode

Shut down

- Select **Shut down**

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

Shut down

- Select **Shut down**

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

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

> The control shuts down.

> Once shutdown has concluded, the control displays the text **Now you can switch off.**

- Switch off the main power switch of the machine

5

Status Displays

5.1 Overview

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

The control offer the following status displays:

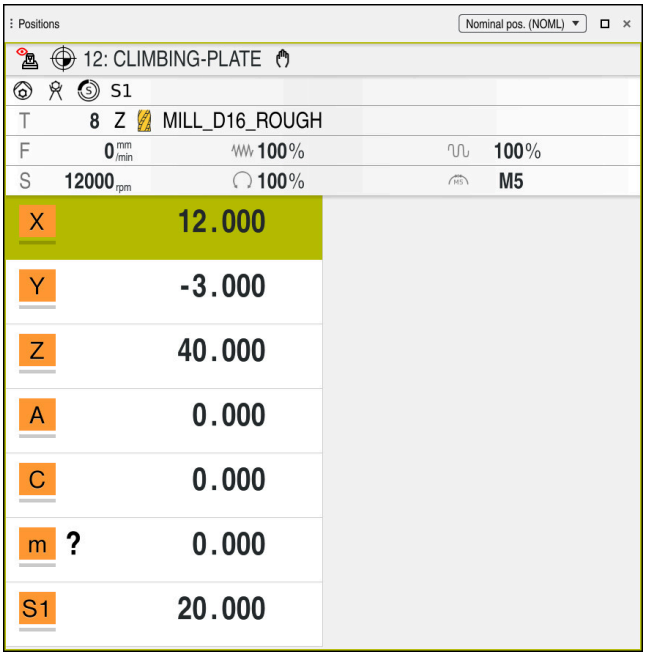
- General status display and position display in the **Positions** workspace
  Further information: "Positions workspace", Page 109
- Status overview on the TNC bar
  Further information: "Status overview on the TNC bar", Page 115
- Additional status displays for specific areas in the **Status** workspace
  Further information: "Status workspace", Page 117
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
  Further information: "Simulation status workspace", Page 131

5.2 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



Positions workspace with general status display

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

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

Further information: "Overview of operating modes", Page 79

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

- Symbols of active and inactive functions, such as Dynamic Collision Monitoring (DCM, option 40)
- 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 486

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
ACTL	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 133
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 114
?	The axis is not referenced.
⚠	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 487
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.
F_{SOS} 0 mm/min	Feed rate when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 483
S_{SOS} 0 U/min	Spindle when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 483

Presets and technology values

Symbol	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 213
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. ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 90 If a feed rate limit is active using the **F MAX** button, the area is called **FMAX** instead of **F**. The control displays the text **FMAX** and the feed rate value in orange. **Further information:** "Feed rate limit FMAX", Page 372
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:** "Program Run operating mode", Page 368
	**RL** tool radius compensation is active. **Further information:** User's Manual for Programming and Testing
	**RR** tool radius compensation is active. **Further information:** User's Manual for Programming and Testing These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 379
	**R+** tool radius compensation is active. **Further information:** User's Manual for Programming and Testing
	**R-** tool radius compensation is active. **Further information:** User's Manual for Programming and Testing These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 379
	3D tool compensation is active. **Further information:** User's Manual for Programming and Testing This icon is transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 379
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 215
	The basic rotation will be taken into account while moving the axes. **Further information:** "Basic rotation selection item", Page 222
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 215

Icon	Meaning
	The tilted working plane will be taken into account while moving the axes. **Further information:** User's Manual for Programming and Testing **Further information:** "3D ROT selection item", Page 223
	The **Tool axis** function is active. **Further information:** "Tool axis selection item", Page 223
	Either the **TRANS MIRROR** function or Cycle **8 MIRRORING** is active. The axes programmed in the function or cycle are mirrored and moved. **Further information:** User's Manual for Machining Cycles **Further information:** User's Manual for Programming and Testing
	The pulsing spindle speed function **S-PULSE** is active. **Further information:** User's Manual for Programming and Testing
	The **PARAXCOMP DISPLAY** function is active.
	The **PARAXCOMP MOVE** function is active. **Further information:** User's Manual for Programming and Testing
	The **PARAXMODE** function is active. This icon might be superimposed on the icons for **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE**. **Further information:** User's Manual for Programming and Testing
TCPM	Either the **M128** function or **TCPM FUNCTION** (option 9) is active. **Further information:** User's Manual for Programming and Testing
	The turning mode **FUNCTION MODE TURN** (option 50) is active. **Further information:** User's Manual for Programming and Testing
	The grinding mode **FUNCTION MODE GRIND** (option 156) is active. **Further information:** User's Manual for Programming and Testing
	Dressing mode (option 156) is active. **Further information:** User's Manual for Programming and Testing
	Dynamic Collision Monitoring (DCM, option 40) is active.

Icon	Meaning
	Dynamic Collision Monitoring (DCM, option 40) is not active. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 226
	Adaptive Feed Control (AFC, option 45) is active in teach-in cut mode.
AFC	Adaptive Feed Control (AFC, option 45) is active in closed-loop mode. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 252
ACC	Active Chatter Control (ACC, option 145) is active. **Further information:** "Active Chatter Control (ACC, option 145)", Page 260
	Global Program Settings (GPS, option 44) are active. **Further information:** "Global Program Settings (GPS, option 44)", Page 261
	Process Monitoring (option 168) is active. **Further information:** "Process Monitoring (option 168)", Page 274

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, option 40) is always visible and cannot be configured.

Definition

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

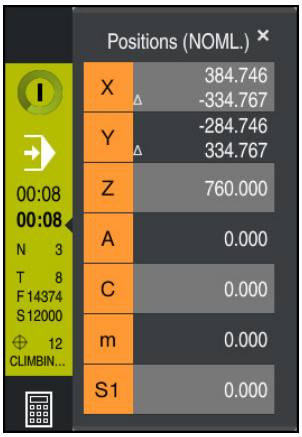
5.3 Status overview on the TNC bar

Application

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

Description of function

General information



When you are machining an NC program or individual NC blocks, the control provides the following information on the TNC bar:

- **Control-in-operation**: current machining status
  Further information: "Definition", Page 116
- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time
  The control displays the run times of the NC program in mm:ss format. As soon as an NC program run time exceeds 59:59, the control changes the format to hh:mm.

  > The control displays the same value for the program run time as on the **PGM** tab of the **Status** workspace.
  >
  > In the **Status** workspace the control shows the program run time in hh:mm:ss format.
  >
  > **Further information:** "Display of the program run time", Page 132

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

Position display

If you select the status overview area, the control opens or closes the position display with the current axis positions. The control uses the same position display mode as in the **Positions** workspace, for example **Actual pos. (ACT)**.

Further information: "Positions workspace", Page 109

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

Press the **actual position capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard. During programming, you can thus transfer the values directly into a programming dialog.

Definition

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

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

Further information: "Interrupting, stopping or canceling program run", Page 373

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

5.4 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

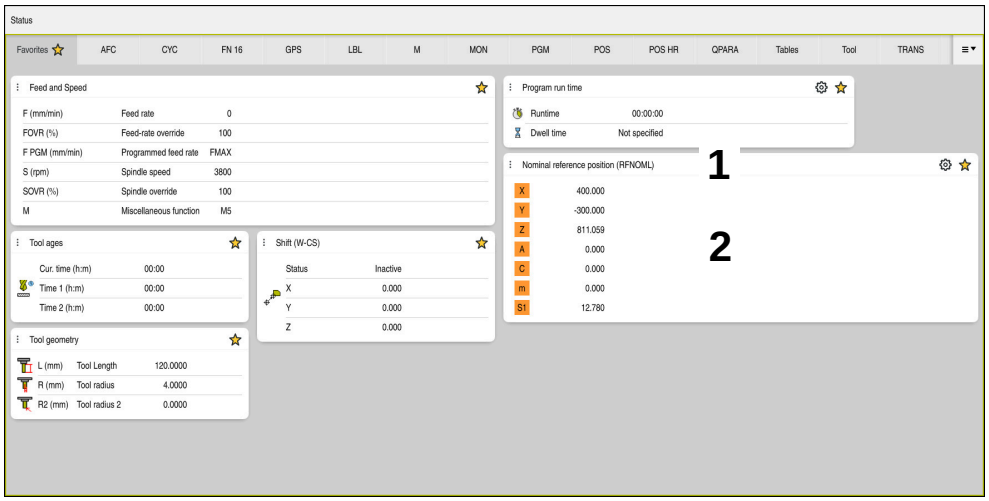
You can open the **Status** workspace in the following operating modes:

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

Further information: "Overview of operating modes", Page 79

Favorites tab

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



Favorites tab

1 Area
2 Contents

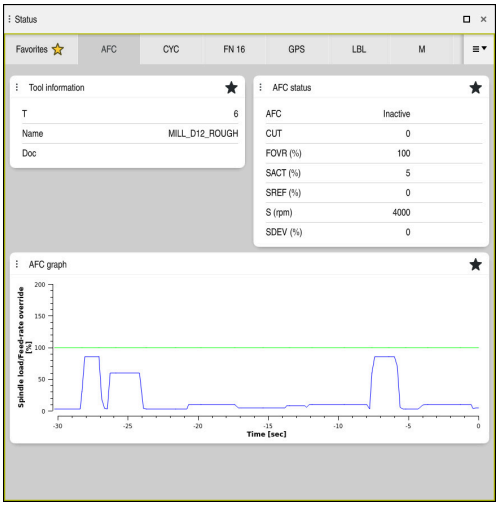
Every area in the status display has a **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

Further information: "Icons on the control's user interface", Page 91

AFC tab (option 45)

The control displays information about the Adaptive Feed Control function (AFC, option 45) on the **AFC** tab.

Further information: "Adaptive Feed Control (AFC, option 45)", Page 252



AFC tab

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Comment about the tool from the tool management
AFC status	■ **AFC** If AFC is being used to control the feed rate, then **Control** is displayed in this area. If the control is not controlling the feed rate, then **Inactive** is displayed in this area. ■ **CUT** Counts the quantity of cuts that have been performed with **FUNCTION AFC CUT BEGIN**, starting from zero. ■ **FOVR (%)** Active factor of the feed-rate potentiometer in percent ■ **SACT (%)** Current spindle load in percent ■ **SREF (%)** Reference load of the spindle in percent Define the reference load of the spindle in the syntax element **LOAD** of the **FUNCTION AFC CUT BEGIN** function. **Further information:** "NC functions for AFC (option 45)", Page 255 ■ **S (rpm)** Spindle shaft speed in rpm ■ **SDEV (%)** Current deviation of the speed in percent

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

CYC tab

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

Area	Contents
Active cycle definition	When you use the **CYCLE 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 TOLERANCE** that are restricted by Dynamic Collision Monitoring (DCM, option 40)

The machine manufacturer uses Dynamic Collision Monitoring (DCM, option 40) to define the restriction of the tolerance.

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

If the tolerance is restricted by DCM, the control displays a gray warning triangle as well as the restricted values.

FN16 tab

On the **FN16** tab the control displays the contents of a file that was output with **FN 16: F-PRINT**.

Further information: User's Manual for Programming and Testing

Area	Contents
Output	Contents of an output file that was output with **FN 16: F-PRINT**, such as measured values or texts.

GPS tab (option 44)

The control displays information about the Global Program Settings (GPS, option 44) on the **GPS** tab.

Further information: "Global Program Settings (GPS, option 44)", Page 261

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 204
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 206
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.

LBL tab

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

Further information: User's Manual for Programming and Testing

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

M tab

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

Further information: User's Manual for Programming and Testing

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.

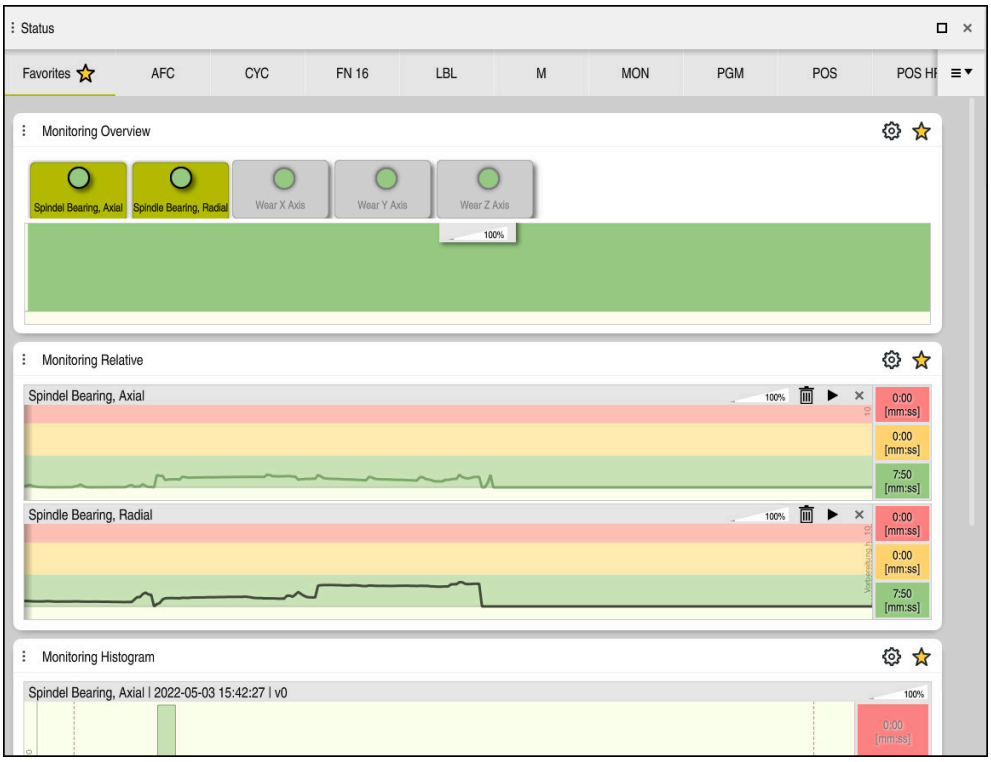
MON tab (option 155)

On the **MON** tab the control shows information about machine components defined to be monitored through Component Monitoring (option 155).

Further information: User's Manual for Programming and Testing

Refer to your machine manual.

The machine manufacturer specifies which machine components are monitored, and to what extent.



MON tab with configured spindle speed monitoring

Area	Contents
Monitoring Overview	The control displays the machine components defined for monitoring. By selecting a component, you hide or show whether it is being monitored.
Monitoring Relative	The control displays the monitoring information for the components being shown in the **Monitoring Overview** area. ■ Green: component works under conditions defined as safe ■ Yellow: component works under warning zone conditions ■ Red: component is overloaded In the **Display settings** window you can select which component the control shows.
Monitoring Histogram	The control shows a graphical evaluation of previous monitoring sessions.

Use the **Settings** symbol to open the **Display settings** window. You can define the height of each graphical depiction for each area.

PGM tab

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

Area	Contents
Parts counter	■ **Quantity** Actual value and nominal value of the parts counter defined with the **FUNCTION COUNT** function **Further information:** User's Manual for Programming and Testing
Program run time	■ **Runtime** Run time of the NC program in hh:mm:ss format ■ **Dwell time** Countdown of the waiting time in seconds from the following functions: ■ **FUNCTION DWELL** ■ Cycle **9 DWELL TIME** ■ Parameter **Q210 DWELL TIME AT TOP** ■ Parameter **Q211 DWELL TIME AT DEPTH** ■ Parameter **Q255 DWELL TIME** **Further information:** "Display of the program run time", Page 132
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

POS tab

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

Area	Contents
Position display, e.g., **Actual reference position (RFACTL)**	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ **Nominal pos. (NOML)** ■ **Actual pos. (ACT)** ■ **Nominal reference position (RFNOML)** ■ **Actual reference position (RFACTL)** ■ **Servo lag (LAG)** ■ **Handwheel superimposed (M118)** **Further information:** "Position displays", Page 133
Feed and Speed	■ Active **Feed** in mm/min If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the **FMAX** button, the control displays **MAX** in square brackets. **Further information:** "Feed rate limit FMAX", Page 372 If the feed rate is limited using the **F limited** button, the control displays the active safety function in square brackets. **Further information:** "Safety functions", Page 482 ■ Active **Feed-rate override** in % ■ Active **Rapid-traverse override** in % ■ Active **Programmed feed rate** in mm/min ■ Active **Spindle speed** in rpm ■ Active **Spindle override** in % ■ Active **Miscellaneous function** in reference to the spindle, such as **M3**
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** User's Manual for Programming and Testing If axis angles are active, the control displays in this area only the values of the physically present axes. The defined values are in the window **3-D rotation** **Further information:** "3D ROT selection item", Page 223
OEM transformation	The machine manufacturer can define an OEM transformation for special turning kinematics. **Further information:** "Definitions", Page 130
Basic transformations	In this area the control shows the values of the active workpiece preset and active transformations in linear and rotary axes, such as a transformation in the X axis with the function **TRANS DATUM**. **Further information:** "Preset management", Page 213

Area	Contents
Special turning transformations	Transformations relevant for turning operations (option 50), such as a 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

POS HR tab

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

Area	Contents
Coordinate system	■ **Machine (M-CS)** With **M118** the handwheel superimpositioning is always in effect in the machine coordinate system **M-CS**. **Further information:** User's Manual for Programming and Testing **i** With the Global Program Settings (GPS, option 44) the coordinate system can be chosen. **Further information:** "Global Program Settings (GPS, option 44)", Page 261
Handwheel superimp.	■ **Max. val.** Maximum value of the individual axes programmed in **M118** or in the **GS** workspace ■ **Actl.val.** Current superimpositioning

QPARA tab

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

Further information: User's Manual for Programming and Testing

You use the **Parameter list** window to define which variables the control shows in the areas.

Further information: "Defining the contents of the QPARA tab", Page 136

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

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 ■ Preset table ■ Datum table ■ Pocket table ■ Touch probe table ■ Grinding tool table ■ Dressing tool table

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:** User's Manual for Programming and Testing
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** User's Manual for Programming and Testing **Further information:** User's Manual for Machining Cycles
Active angle of rotation	Rotation angle defined with either the **TRANS ROTATION** function or Cycle **10 ROTATION** **Further information:** User's Manual for Programming and Testing **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:** User's Manual for Programming and Testing
Center coordinates	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** User's Manual for Machining Cycles

Area	Contents
Active scaling factors	Scaling factors that were defined for the individual axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** User's Manual for Programming and Testing **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** (option 50) **Further information:** User's Manual for Programming and Testing
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:** User's Manual for Programming and Testing

TT tab

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

Further information: "Hardware enhancements", Page 76

Area	Contents
TT Tool to be measured	■ **T** Tool number ■ **Name** Tool name ■ **Measure what?** Measurement method chosen for tool measurement, such as **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 (option 50), in this area the control shows the smallest measured tipping angle. The value of the angle can also be negative. **Further information:** "Definitions", Page 130 ■ **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 Tool cutting edges	**Number** List of the measurements performed and the measured values of the individual cutting edges

Tool tab

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

Further information: "Tool types", Page 168

Contents for dressing, milling, and grinding tools (option 156)

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Note on the tool
Tool geometry	■ **L** Tool length ■ **R** Tool radius ■ **R2** Corner radius of the tool
Tool allowances	■ **DL** Delta value for the tool length ■ **DR** Delta value for the tool radius ■ **DR2** Delta value for the corner radius of the tool With **Program**, the control displays the values from a tool call with **TOOL CALL** or from a tool compensation with a compensation table ***.tcs**. **Further information:** User's Manual for Programming and Testing With **Table**, the control displays the values from the tool management. **Further information:** "Tool management ", Page 184
Tool ages	■ **Cur. time (h:m)** Time in hours and minutes the tool has been engaged ■ **Time 1 (h:m)** Service life of the tool ■ **Time 2 (h:m)** Maximum service life at tool call
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool
Tool type	■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**)

Other contents for turning tools (option 50)

Area	Contents
Tool geometry	■ **ZL (mm)** Tool length in Z direction ■ **XL (mm)** Tool length in X direction ■ **RS (mm)** Cutter radius ■ **YL (mm)** Tool length in Y direction
Tool allowances	■ **DZL (mm)** Delta value in Z direction ■ **DXL (mm)** Delta value in X direction ■ **DRS (mm)** Delta value for the cutter radius ■ **DCW (mm)** Delta value for the width of the recessing tool
Tool type	■ **Tool Axis** ■ **TO** Tool orientation ■ **Type** Tool type, e.g. **TURN**

Definitions

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

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 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: "Favorites tab", Page 117
- **CYC**
  Further information: "CYC tab", Page 119
- **FN16**
  Further information: "FN16 tab", Page 119
- **LBL**
  Further information: "LBL tab", Page 121
- **M**
  Further information: "M tab", Page 121
- **PGM**
  Further information: "PGM tab", Page 123
- **POS**
  Further information: "POS tab", Page 124
- **QPARA**
  Further information: "QPARA tab", Page 125
- **Tables**
  Further information: "Tables tab", Page 126
- **TRANS**
  Further information: "TRANS tab", Page 126
- **TT**
  Further information: "TT tab", Page 128
- **Tool**
  Further information: "Tool tab", Page 129

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
- Status overview on the control bar
- **PGM** tab of the **Simulation status** workspace
- **Simulation** workspace in the **Editor** operating mode

Modify the **Settings** in the **Program run time** area in order to influence the calculated program run time.

Further information: "PGM tab", Page 123

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 how long the execution of the active NC program will take. During program run the control updates the remaining run time.

Further information: "Tool usage test", Page 191

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 remaining run time is reset by the following:

- Selecting a new NC program for program run
- The **Internal stop** button
- Generating a 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 373

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:

- **Positions** workspace
- Status overview on the control bar
- **POS** tab of the **Status** workspace
- **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:** User's Manual for Programming and Testing

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, for example **Actual pos. (ACT)**
- > The control displays the positions in the selected mode.

Notes

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

  Further information: "Axis display and position display", Page 110

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 shows.

Further information: "QPARA tab", Page 125

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

- Select the **QPARA** tab
- Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.
- Enter numbers, such as **1,3,200-208**
- 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}.
- For variable texts in QS parameters the control shows the first 30 characters, i.e. the contents might be truncated.

6

Powering On and Off

6.1 Powering on

Application

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

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

Related topics

- Absolute and incremental position encoders
  Further information: "Position encoders and reference marks", Page 153

Description of function

DANGER

Caution: hazard to the user!

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

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

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

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

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

- Absolute position encoders
  If the machine has absolute position encoders, the control is in 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: "Referencing workspace", Page 141
  Further information: "Manual operation application", Page 146

6.1.1 Powering the machine and the control on

To switch the machine on:

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

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

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

  Further information: "Referencing workspace", Page 141

- ▶ 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: "Manual operation application", Page 146

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

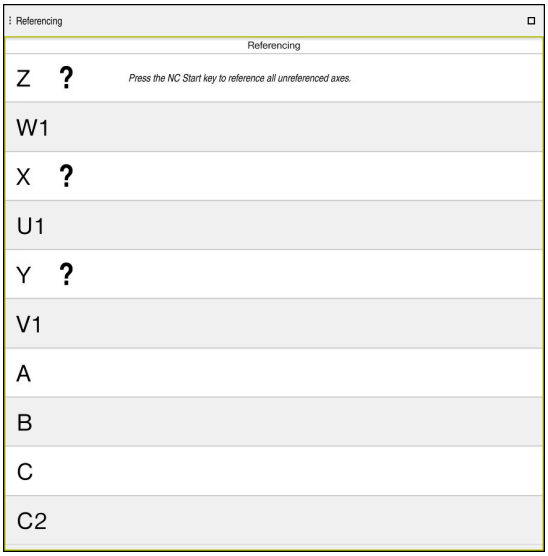
6.2 Referencing workspace

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.



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 **Tilt working plane** function (option 8) was active before the control was shut down, then the control will automatically reactivate this 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: User's Manual for Programming and Testing

6.3 Powering off

Application

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

Description of function

You 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 the program** window. You can save the changes, discard them or cancel the shutdown.

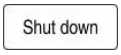
6.3.1 Shutting down the control and powering-off the machine

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 the program** window.

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

> The control shuts down.

> Once shutdown has concluded, 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 on the control can delay the shutdown, such as a connection with **Remote Desktop Manager** (option 133)
  Further information: "Remote Desktop Manager window (option 133)", Page 527

7

Manual Operation

7.1 Manual operation application

Application

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

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 147
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 149

Description of function

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

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

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

Button	Meaning
Handwheel	The control displays this toggle switch if a handwheel is configured for the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic Handwheel", Page 459
M	Define a miscellaneous function **M** or use the selection window to choose one and activate it with the **NC start** key. **Further information:** User's Manual for Programming and Testing
S	Define the spindle speed **S** activate it with the **NC start** key, and also switch on the spindle. **Further information:** User's Manual for Programming and Testing
F	Define the feed rate **F** and activate it with the **OK** button. **Further information:** User's Manual for Programming and Testing
T	Define a tool **T** or use the selection window to choose one and insert it with the **NC start** key. **Further information:** User's Manual for Programming and Testing
3D ROT	The control opens a window for the 3D rotation settings (option 8). **Further information:** User's Manual for Programming and Testing
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:** User's Manual for Programming and Testing
DCM	The control opens the **Collision monitoring (DCM)** window, where you can activate or deactivate Dynamic Collision Monitoring (DCM, option 40). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 230

Button	Meaning
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 486
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 149
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 213

Note

The machine manufacturer defines which miscellaneous functions are available on the control and which are allowed in the **Manual operation** application.

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 331

Related topics

- Programming traverse movements
  Further information: User's Manual for Programming and Testing
- Executing traverse movements in the **MDI** application
  Further information: "The MDI Application ", Page 363

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 459

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

Further information: "Status Displays", Page 107

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

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

Further information: "Status overview on the TNC bar", Page 115

7.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:

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

- Press the axis key of the desired axis

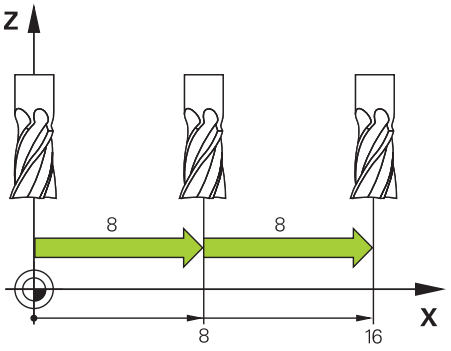
> The control moves the axis as long as you press the key.

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

You can move more than one axis at a time.

7.2.2 Incremental jog positioning of axes

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



To position an axis incrementally:

- Select the **Manual** operating mode
- Select the **Manual operation** application
- Select **Jog increment**

Jog increment

> The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.

- Enter the jog increment for linear axes and rotary axes
- Press the axis key of the desired axis

X+

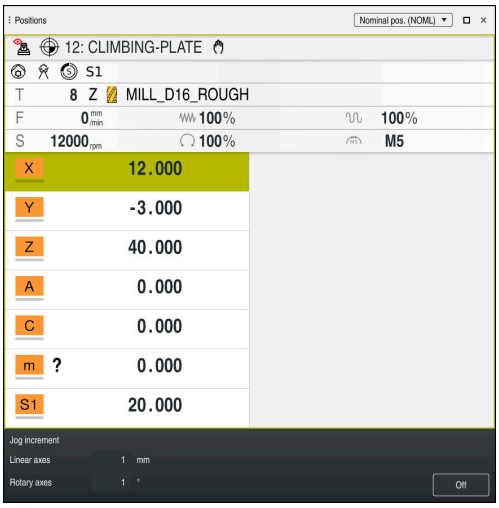
> The control positions the axis in the selected direction by the defined jog increment.

- Select **Jog increment On**

Jog increment On

> The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

> You can also end incremental jog positioning with the **Off** button in the **Jog increment** area.



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

Note

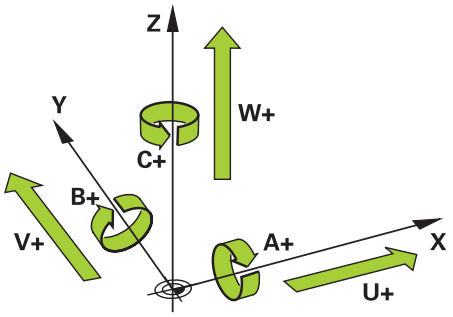
When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

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 on milling machines

The axes **X**, **Y** and **Z** on your milling machine are designated as the main axis (1st axis), secondary axis (2nd axis) and tool axis. The main axis and secondary axis define the working plane.

The axes are associated as follows:

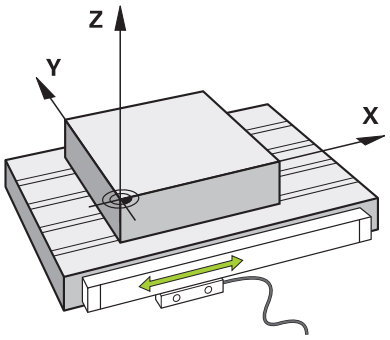
Main axis	Secondary axis	Tool axis	Working plane
X	**Y**	**Z**	**XY**, also **UV**, **XV**, **UY**
Y	**Z**	**X**	**YZ**, also **WU**, **ZU**, **WX**
Z	**X**	**Y**	**ZX**, also **VW**, **YW**, **VZ**

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

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

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 198

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

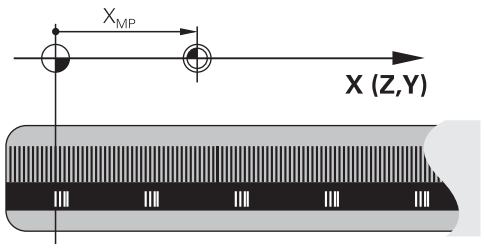
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

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 141



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 159

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 200 If you program **M91** in an NC block, the defined values are referenced to the machine datum. **Further information:** User's Manual for Programming and Testing
 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:** User's Manual for Programming and Testing
	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 153 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 141
	**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 204 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 213 **Further information:** User's Manual for Programming and Testing If no transformations are defined, the entries in the NC program refer to the workpiece preset.

Icon	Preset
	**Workpiece datum** You define the workpiece datum with transformations in the NC program, for example with **TRANS DATUM** or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset. If you tilt the working plane (option 8), the workpiece datum is the point around which the workpiece is rotated.

9

Tools

9.1 Fundamentals

To use the control's functions, you must define the tools for the control using real data (e.g., the radius). This makes programming easier and improves process reliability.

To add a tool to the machine, follow the sequence below:

- Prepare your tool and clamp the tool into a suitable tool holder.
- To measure the tool dimensions, starting from the tool carrier preset, measure the tool (e.g., using a tool presetter). The control needs these dimensions for calculating the paths.

  Further information: "Tool carrier reference point", Page 159
- Further tool data are needed to completely define the tool. Take these tool data from the manufacturer's tool catalog, for example.

  Further information: "Tool data for the tool types", Page 171
- Save all collected tool data of this tool in the tool management.

  Further information: "Tool management ", Page 184
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 188
- After finishing tool definition, program a tool call within an NC program.

  Further information: User's Manual for Programming and Testing
- 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: User's Manual for Programming and Testing
- 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 191
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: User's Manual for Programming and Testing

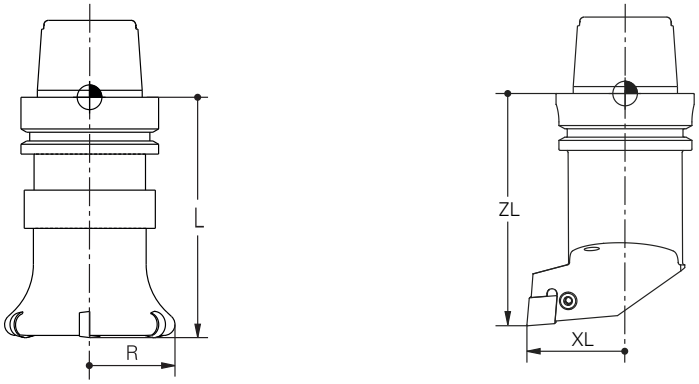
9.2 Presets on the tool

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 154

9.2.1 Tool carrier reference point

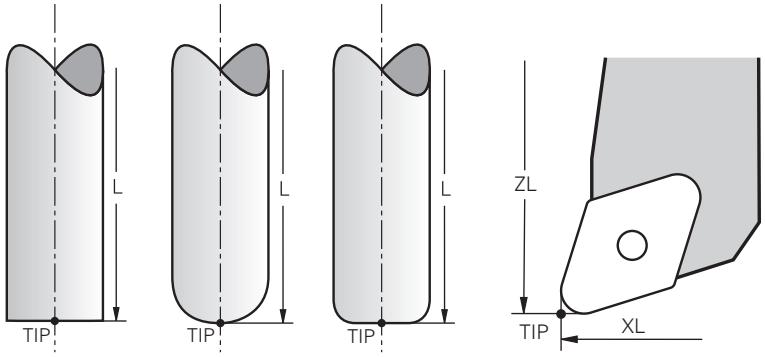


The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

Further information: "Tool management ", Page 184

9.2.2 Tool tip TIP



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

Further information: "Tool coordinate system T-CS", Page 210

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

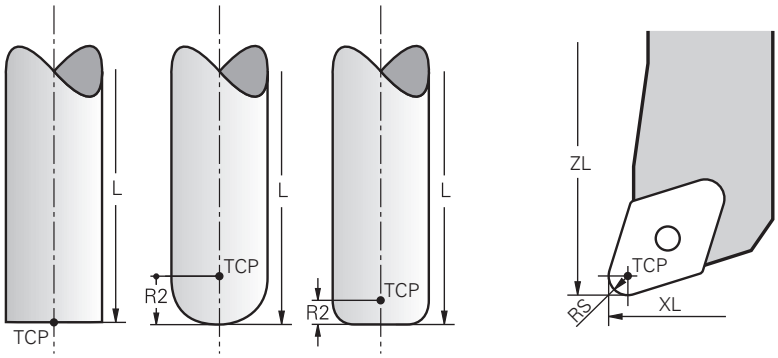
You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool data for the tool types", Page 171

In the case of lathe tools (option 50) the control uses the theoretical tool tip, i. e. the longest measured values **ZL**, **XL** and **YL**.

9.2.3 Tool center point (TCP, tool center point)



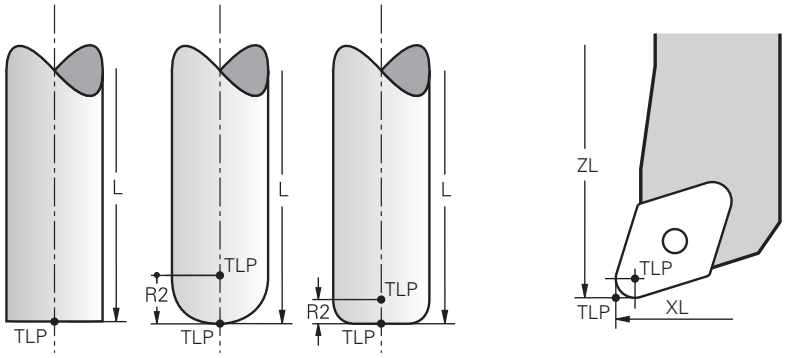
The tool center point is the center of the tool radius **R**. If a second tool radius (**R2**) is defined, the tool center point is offset from the tool tip by this value.

In the case of turning tools (option 50), the tool center point is at the center of cutter radius **RS**.

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

Further information: "Tool data for the tool types", Page 171

9.2.4 Tool location point (TLP, tool location point)

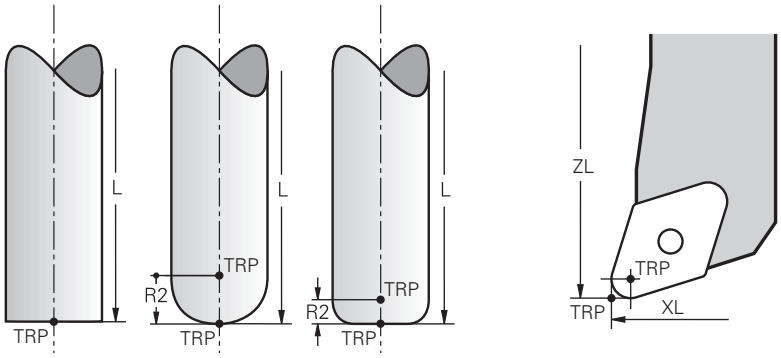


The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

In the **FUNCTION TCPM** function (option 9), you can also choose the tool location point to be at the tool center point.

Further information: User's Manual for Programming and Testing

9.2.5 Tool rotation point (TRP, tool rotation point)



When applying the tilting function with **MOVE** (option 8), the control tilts around the tool rotation point. By default, the tool rotation point is at the tool tip.

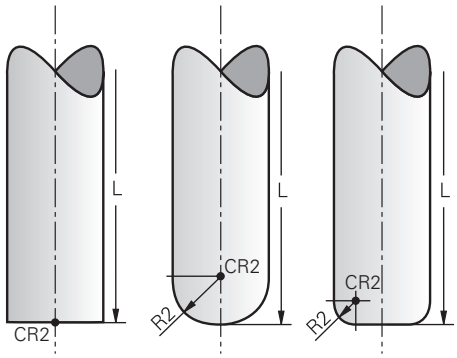
When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: User's Manual for Programming and Testing

In the **FUNCTION TCPM** function (option 9), you can also choose the tool rotation point to be at the tool center point.

Further information: User's Manual for Programming and Testing

9.2.6 Tool radius 2 center (CR2, center R2)



The control uses the tool radius 2 center in conjunction with 3D tool compensation (option 9). 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: User's Manual for Programming and Testing

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

9.3 Tool data

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 184

Description of function

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

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

Further information: User's Manual for Programming and Testing

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 will substitute them by uppercase letters upon saving.

Note

- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks 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 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 a workshop). This allows you to coordinate the tools of multiple machines more easily.

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

Related topics

- **DB_ID** column of tool management
  Further information: "Tool table tool.t", Page 405

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 164

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 sets of tool data can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

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 159

Further information: "Creating an indexed tool", Page 165

Examples of an application of indexed tools:

- Step drill
  The tool data of the main tool contain the drill tip, which corresponds to the maximum length. The tool steps are defined are indexed tools. This makes the lengths equal the actual tool dimensions.
- NC center drill
  The main tool is used for defining the theoretical tool tip as the maximum length. This can be used for centering, for example. The indexed tool defines a point along the tool tooth. This can be used for deburring, for example.
- Cut-off milling cutter or T-slot milling cutter
  The main tool is used for defining the lower point of the cutting edge, which equals the maximum length. The indexed tool defines the upper point of the cutting edge. When using the indexed tool for cutting-off, the specified workpiece height can be directly programmed.

Creating an indexed tool

To create an indexed tool:

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

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

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

- Press **OK**
- The control adds table row **5**.
- Define all necessary tool data including the maximum tool length.

  Further information: "Tool data for the tool types", Page 171

- Select **Insert tool**
- The control opens the **Insert tool** pop-up window.
- Define the tool type
- Define the tool number of the indexed tool (e.g., **T5.1**)

> The main tool number and an index after the dot define an indexed tool.

- Press **OK**
- The control adds table row **5.1**.
- Define all required tool data

  Further information: "Tool data for the tool types", Page 171

> The control does not adopt any main tool data!
>
> 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 159

Notes

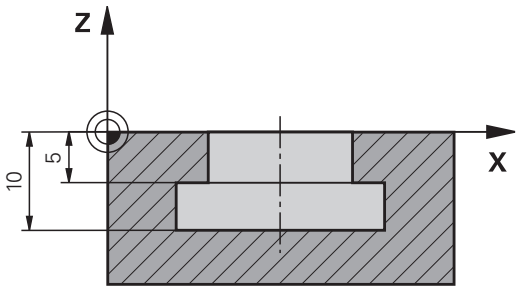
- The control describes some parameters automatically (e.g., the current tool age **CUR_TIME**). The control describes these parameters separately for each table row.

  Further information: "Tool table tool.t", Page 405
- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.
- Up to nine indexed tools can be added to each main tool.

  When defining 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 ensures, for example, that the main tool can still be used.

  Further information: User's Manual for Programming and Testing

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 tool data are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table lines.

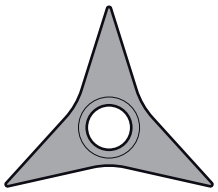
The T-slot is programmed in two machining steps:

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

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

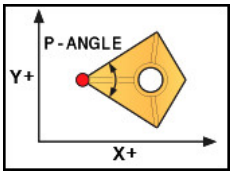
Example of a FreeTurn tool

You need the following tool data for a FreeTurn tool:



FreeTurn tool with three finishing teeth

Integrating information about the point angles **P-ANGLE** and the tool length **ZL**, e.g. **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 159
 **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 159
 **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:** "Subgroups of technology-specific tool types", Page 169
 **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**	Cutting-edge length	You can get the tooth length **CUTLENGTH** from the tool catalog.
	Toolcarrier kinematics	Using the optional tool-carrier kinematics, the control can monitor the tool for collisions, for example. Assign the same kinematics to each single tooth.

9.3.5 Tool types

Application

Depending on the selected tool type, the control displays the editable tool data in the tool management.

Related topics

- Editing the tool data in the tool management
  Further information: "Tool management ", Page 184

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected in the **TYPE** column of the tool management:

Icon	Tool type	Number
	Milling cutter (**MILL**)	0
	Rough cutter (**MILL_R**)	9
	Finishing cutter (**MILL_F**)	10
	Face mill (**MILL_FACE**)	14
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Chamfer mill (**MILL_CHAMFER**)	24
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4

Icon	Tool type	Number
	Turning tool (**TURN**) **Further information:** "Types within the turning tools", Page 170	29
	Touch probe (**TCHP**)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5
	Piloted counterbore (**TSINK**)	6
	Boring tool (**BOR**)	7
	Back boring tool (**BCKBOR**)	8
	Thread miller (**GF**)	1
	Thread miller with chamfer (**GSF**)	16
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19
	Circular thread mill (**ZBGF**)	20
	Grinding wheel (**GRIND**) **Further information:** "Types within the grinding tools", Page 170	30
	Dressing tool (**DRESS**) **Further information:** "Types within the dressing tools", Page 170	31

These tool types allow filtering the tools in the tool management.

Further information: "Tool management ", Page 184

Subgroups of technology-specific tool types

In the **TYPE** column of the tool management, a technology-specific tool type can be defined, depending on the selected tool type. The control offers the **TYPE** column for the **TURN**, **GRIND** and **DRESS** tool types. Specify the tool type more precisely within these technologies.

Types within the turning tools

Select between the types below within the turning tools:

Icon	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

Types within the grinding tools

Select between the types below within the grinding tools:

Icon	Tool type	Number
	Cylindrical grinding pin (**GRIND_PIN**)	1
	Conical grinding pin (**GRIND_CONE**)	2
	Cup wheel (**GRIND_CUP**)	3
	Straight wheel (**GRIND_CYLINDER**) Currently no function	26
	Slant wheel (**GRIND_ANGULAR**) Currently no function	27
	Facing wheel (**GRIND_FACE**) Currently no function	28

Types within the dressing tools

Select between the types below within the dressing tools:

Icon	Tool type	Number
	Stationary dresser with radius (**DRESS_FIX_RADIUS**)	101
	Horn-type dresser (**HORNED**) Currently no function	102
	Rotating dresser with radius (**DRESS_ROT_RADIUS**)	103
	Stationary dresser (flat) (**DRESS_FIX_FLAT**)	110
	Rotating (flat) (**DRESS_ROT_FLAT**)	120

9.3.6 Tool data for the tool types

Application

The tool data provide the control with all information necessary for calculating and checking the required movements.

The necessary data depend on the technology and the tool type.

Related topics

- Editing the tool data in the tool management
  Further information: "Tool management ", Page 184
- Tool types
  Further information: "Tool types", Page 168

Description of function

Some of the necessary tool data can be determined using the following options:

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

In the tables below, the relevance of the parameters is sub-divided into the optional, recommended and required categories.

The control takes recommended parameters into account for at least one of the functions below:

- Simulation
  Further information: User's Manual for Programming and Testing
- Machining or touch probe cycles
  Further information: User's Manual for Machining Cycles
  Further information: User's Manual for Measuring Cycles for Workpieces and Tools
- Dynamic Collision Monitoring (DCM, option 40)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226

Tool data for milling and drilling tools

The control offers the following parameters for milling and drilling tools:

Icon and parameter	Meaning	Intended use
 **L**	Length	Required for all milling and drilling tool types
 **R**	Radius	Required for all milling and drilling tool types
 **R2**	Radius 2	Required for the following milling and drilling tool types: ■ **Ball-nose cutter** ■ **Toroid cutter**
 **DL**	Delta value of length	Optional The control describes this parameter in connection with touch probe cycles.
 **DR**	Delta value of radius	Optional The control describes this parameter in connection with touch probe cycles.
 **DR2**	Delta value of radius 2	Optional The control describes this parameter in connection with touch probe cycles.
 **LCUTS**	Tooth length	Recommended
 **RCUTS**	Tooth width	Recommended
 **LU**	Useful length	Recommended
 **RN**	Neck radius	Recommended
 **ANGLE**	Plunge angle	Recommended for the following milling and drilling tool types: ■ **Milling tool** ■ **Roughing mill** ■ **Finishing cutter** ■ **Ball-nose cutter** ■ **Toroid cutter**

Icon and parameter	Meaning	Intended use
 **PITCH**	Thread pitch	Recommended for the following milling and drilling tool types: ■ **Tapping tools** ■ **Thread mill** ■ **Thread miller with chamfer** ■ **Thread mill with single thread** ■ **Thread mill w/ indexable insert** ■ **Thread drilling/milling cutter** ■ **Circular thread mill**
 **T-ANGLE**	Point angle	Recommended for the following milling and drilling tool types: ■ **Drill** ■ **NC center drill** ■ **Countersink** ■ **Chamfer cutter**
 **NMAX**	Maximum spindle speed	Optional
R_TIP	Radius at the tip	Recommended for the following milling and drilling tool types: ■ **Face mill** ■ **Countersink** ■ **Chamfer cutter**

- All tool types listed in the **TYP** column are milling and drilling tools except for:
  - **Touch probe**
  - **Turning tool**
  - **Grinding wheel**
  - **Dressing tool**

  Further information: "Tool types", Page 168
- The parameters are described in the tool table.

  Further information: "Tool table tool.t", Page 405

Tool data for turning tools (option 50)

The control offers the following parameters for turning tools:

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	Required for all turning tool types
 **XL**	Tool length 2	Required for all turning tool types
 **YL**	Tool length 3	Required for all turning tool types
 **RS**	Cutting radius	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Recessing tool** ■ **Recess-turning tool**
 **TYPE**	Lathe tool type	Required for all turning tool types
 **TO**	Tool orientation	Required for all turning tool types Depending on the selected **TYPE** tool type, the control shows selected tool orientations with different graphics. The machine manufacturer can change this assignment.
 **DZL**	Delta value of tool length 1	Optional The control describes this value in connection with touch probe cycles.
 **DXL**	Delta value of tool length 2	Optional The control describes this value in connection with touch probe cycles.
 **DYL**	Delta value of tool length 3	Optional The control describes this value in connection with touch probe cycles.
 **DRS**	Delta value of cutter radius	Optional The control describes this value in connection with touch probe cycles.
 **DCW**	Delta value of cutter width	Optional The control describes this value in connection with touch probe cycles.

Icon and parameter	Meaning	Intended use
 **ORI**	Angle of orientation	Required for all turning tool types
 **T-ANGLE**	Tool angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
 **P-ANGLE**	Point angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
  **CUTLENGHT**	Cutting-edge length	Recommended
  **CUTWIDTH**	Tooth width	Required for the turning tool types below: ■ **Recessing tool** ■ **Recess-turning tool** Recommended for the other turning tool types
 **SPB-INSERT**	Angular offset	Required for all turning tool types

- The **TYP** column of the **Turning tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define turning tools.
  Further information: "Tool types", Page 168
  Further information: "Types within the turning tools", Page 170
- The parameters are described in the turning tool table.
  Further information: "Turning tool table toolturn.trn (option 50)", Page 415

Tool data for grinding tools (option 156)

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 control offers the following parameters for grinding tools:

Icon and parameter	Meaning	Intended use
 **TYPE**	Grinding tool type	Required for all grinding tool types
 **R-OVR**	Radius	Required for all grinding tool types This value must not be edited after initial dressing.
 **L-OVR**	Overhang	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel** This value must not be edited after initial dressing.
 **LO**	Overall length	Required for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin** This value must not be edited after initial dressing.
 **LI**	Length to the inner edge	Required for the **Conical grinding pin** grinding tool type This value must not be edited after initial dressing.

Icon and parameter	Meaning	Intended use
 **B**	Width	Required for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Cup wheel** This value must not be edited after initial dressing.
 **G**	Depth of grinding tool	Required for the **Cup wheel** grinding tool type This value must not be edited after initial dressing.
ALPHA	Slant angle	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel** For the **Cup wheel** grinding tool type, you must define the angle 90°.
GAMMA	Corner angle	Required for the grinding tool types below: ■ **Conical grinding pin** ■ **Cup wheel**
 **RV**	Radius at the edge for **L-OVR**	Optional for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin**
 **RV1**	Radius at the edge for **LO**	Optional for the grinding tool types below: ■ **Cylindrical grinding pin** ■ **Conical grinding pin**
 **RV2**	Radius at the edge for **LI**	Optional for the **Conical grinding pin** grinding tool type
 **HWI**	Angle for a relief cut on the inner edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
 **HWA**	Angle for a relief cut on the outer edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
COR_TYPE	Selection of compensation method	Required for all grinding tool types
INIT_D_OK	Initial dressing	Currently no function
MESS_OK	Measuring the grinding tool	The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**.
T-DRESS	Tool number of the dresser	The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Corresponds to parameter **A_NR_D** in the grinding tool table

Icon and parameter	Meaning	Intended use
 **dR-OVR**	Delta value of radius	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
 **dL-OVR**	Delta value of overhang	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
 **dLO**	Delta value of total length	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
 **dLI**	Delta value of length up to the inner edge	The control uses this parameter only with the **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** selection in the **COR_TYPE** parameter.
 **DRESS-N-D**	Default value of diameter dressing counter	Currently no function
 **DRESS-N-A**	Default value of outer edge dressing counter	Currently no function Optional
 **DRESS-N-I**	Default value of inner edge dressing counter	Currently no function Optional
 **DRESS-N-D-ACT**	Diameter dressing counter	Currently no function
 **DRESS-N-A-ACT**	Outer edge dressing counter	Currently no function
 **DRESS-N-I-ACT**	Inner edge dressing counter	Currently no function
 **R_SHAFT**	Radius of the tool shank	Optional
 **R_MIN**	Min. permissible radius	Optional
 **B_MIN**	Min. permissible width	Optional

Icon and parameter	Meaning	Intended use
 V_MAX	Maximum permissible cutting speed	Optional
 AD	Retraction amount at the diameter	Required for all grinding tool types
 AA	Retraction amount at the outer edge	Required for all grinding tool types
 AI	Retraction amount at the inner edge	Required for all grinding tool types

- The **TYP** column of the **Grinding wheel** tool type as well as the associated technology-specific tool types in the **TYPE** column define grinding tools.
  Further information: "Tool types", Page 168
  Further information: "Types within the grinding tools", Page 170
- The parameters are described in the grinding tool table.
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 419

Tool data for dressing tools (option 156)

The control offers the following parameters for dressing tools:

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	Required for dressing tool types
 **XL**	Tool length 2	Required for all dressing tool types
 **YL**	Tool length 3	Required for all dressing tool types
 **RS**	Cutting radius	Required for the dressing tool types below: ■ **Stationary dresser with radius** ■ **Rotating dresser with radius**
CUTWIDTH	Width of tooth	Required for the dressing tool types below: ■ **Stationary dresser (flat)** ■ **Rotating dresser (flat)**
 **TYPE**	Dressing tool type	Required for all dressing tool types
 **TO**	Tool orientation	Required for all dressing tool types
 **DZL**	Delta value of tool length 1	Optional
 **DXL**	Delta value of tool length 2	Optional
 **DYL**	Delta value of tool length 3	Optional
 **DRS**	Delta value of cutter radius	Optional
N-DRESS	Tool speed	Required for the dressing tool types below: ■ **Rotating dresser with radius** ■ **Rotating dresser (flat)**

- The **TYP** column of the **Dressing tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define dressing tools.
  Further information: "Tool types", Page 168
  Further information: "Types within the dressing tools", Page 170
- The parameters are described in the dressing tool table.
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 428

Tool data for touch probes

NOTICE

Danger of collision!

The control cannot use Dynamic Collision Monitoring (DCM) to protect L-shaped styluses against collisions. When using a touch probe with an L-shaped stylus there is a risk of collision!

- Carefully run in the NC program or program section in the **Program Run Single Block** operating mode
- Watch out for possible collisions!

The control offers the following parameters for touch probes:

Icon and parameter	Meaning	Intended use
 **L**	Length	Required
 **R**	Radius	Required
TP_NO	Number in the touch probe table	Required
 **TYPE**	Type of touch probe	Required
 **F**	Probing feed rate	Required
 **FMAX**	Rapid traverse in probing cycle	Optional
 **F_PREPOS**	Pre-positioning at rapid traverse	Required
 **TRACK**	Orienting the touch probe in each probing process	Required When selecting **L-TYPE** in the **STYLUS** parameter, **ON** must be selected
 **REACTION**	Trigger **NCSTOP** or **EMERGSTOP** in case of collision	Required
 **SET_UP**	Set-up clearance	Recommended

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

- The **TYP** column of the **Touch probe** tool type as well as the touch probe model in the **TYPE** column define touch probes.
  Further information: "Tool types", Page 168
- The parameters are described in the touch probe table.
  Further information: "Touch probe table tchprobe.tp", Page 431

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 adding tools, editing tool data and deleting tools.

Related topics

- Creating new tools
  Further information: "Configuring a tool", Page 97
- Table workspace
  Further information: "Table workspace", Page 396
- Form workspace
  Further information: "Form workspace for tables", Page 403

Description of function

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

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

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 405
- Turning tool table **toolturn.trn** (option 50)
  Further information: "Turning tool table toolturn.trn (option 50)", Page 415
- Grinding tool table **toolgrind.grd** (option 156)
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 419
- Dressing tool table **tooldress.drs** (option 156)
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 428
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 431

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

Further information: "Pocket table tool_p.tch", Page 435

Tool data can be edited in the **Table** workspace or in the **Form** workspace. In the **Form** workspace the control shows the correct tool data for each tool type.

Further information: "Tool data", Page 163

Notes

- When creating a new tool, the length **L** and radius **R** columns are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.
- The tool data of tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.
- When editing tool data, bear in mind that the current tool may have been entered in column **RT** as a replacement tool of another tool!
- 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.

9.4.1 Importing and exporting tool data

Application

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

Description of function

The control transmits tool data as a CSV file.

Further information: User's Manual for Programming and Testing

The tool data transfer file is structured as follows:

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

The column names and the tool data stand between double quotation marks and are separated by semicolons.

Please note the following regarding the transfer file:

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

Importing tool data

To import tool data:

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

- Activate **Edit**
- > The control enables tool management for editing.

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

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

NOTICE

Caution: Data may be lost!

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

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

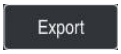
Exporting tool data

To export tool data:

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

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

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

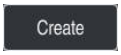


- Select **Export**
- The control opens the **Save as** window.
- Select a path

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

- Enter the file name
- Select the file type

> Select either **TNC7 (*.csv)** or **TNC 640 (*.csv)**. The internal formatting of the transfer files differs. If you wish to use the data in a previous control model, select **TNC 640 (*.csv)**.



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

Notes

NOTICE

Caution: Possible material damage!

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

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

- The transfer file must be saved under **TNC:\table**.
- The internal formatting of the transfer files differs:
  - **TNC7 (*.csv)** writes the values between double quotation marks and separates the values by semicolons
  - **TNC 640 (*.csv)** writes the values partly between brackets and separates the values by commas

  The TNC7 can both import and export both transfer files.

9.5 Tool carrier management

Application

Tool carrier management allows parameterizing and assigning tool carriers.

The control represents the tool carriers graphically in the simulation and takes the tool carriers into account by calculation, e. g. in Dynamic Collision Monitoring (DCM, option 40).

Related topics

- **Simulation** workspace
  Further information: User's Manual for Programming and Testing
- Dynamic Collision Monitoring (DCM, option 40)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226

Description of function

To ensure that the control takes the tool carriers into account in its calculations and in the display:

- Save the tool carrier or tool carrier templates
- Parameterize the tool carrier templates
  Further information: "Parameterizing tool carrier templates", Page 190
- Assign a tool carrier
  Further information: "Assigning a tool carrier", Page 190

If you are using M3D or STL files instead of tool carrier templates, you can assign the files directly to the tools. The parameterization step is superfluous here.

Tool carriers in STL format must meet the following requirements:

- Max. 20 000 triangles
- Triangular mesh forms a closed shell

If an STL file does not meet the requirements of the control, then the control issues an error message.

For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.

Further information: "Options for fixture files", Page 233

Tool-carrier templates

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 changeable dimensions.

Tool carrier templates must be stored as files with a ***.cft** file name extension in the **TNC:\system\Toolkinematics** directory.

They can be downloaded through the following link:

http://www.klartext-portal.com/nc-solutions/en

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

The tool carrier templates can be parametrized in the **ToolHolderWizard** window. This defines the tool carrier dimensions.

Further information: "Parameterizing tool carrier templates", Page 190

The parametrized tool carriers with the ***.cfx** extension are saved at **TNC:\system\Toolkinematics**.

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

Icon	Function
	Close the application
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Restore initial view
	Select alignment, e.g. plan view

9.5.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

- Select the **Files** operating mode
- Open the **TNC:\system\Toolkinematics** folder
- Double-tap or -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 **Quit**

9.5.2 Assigning a tool carrier

To assign a tool carrier to a tool:

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

- Activate **Edit**
- In the **Spec. functions** panel, select the **KINEMATIC** parameter
- The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- Select the desired tool carrier

- Select **OK**
- The control assigns the tool carrier to the tool.

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

  For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.

  Further information: "Fixture monitoring (option 40)", Page 232

Notes

- In the simulation, the tool carriers can be checked for collisions with the workpiece.

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

  HEIDENHAIN recommends machining with tool axis **Z**. Using Advanced Functions Set 1 (software option 8), you can tilt the working plane to the angle of exchangeable angle heads and continue working with tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM, option 40). This enables the tool carriers to protect against collisions with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226
- A grinding tool that is to be dressed must not contain any tool carrier kinematics (option 156).

9.6 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 438
- Tool usage test in Batch Process Manager (option 154)

  Further information: User's Manual for Programming and Testing

Requirement

- A tool usage file is needed for performing a tool usage test

  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.

  Further information: "Tool usage file", Page 438
- The **Generate tool-usage file** function setting is set to either **once** or **always**

  Further information: "Channel settings", Page 494
- Use the same tool table for the simulation as for the program run

  Further information: User's Manual for Programming and Testing

Description of function

Creating the tool usage file

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

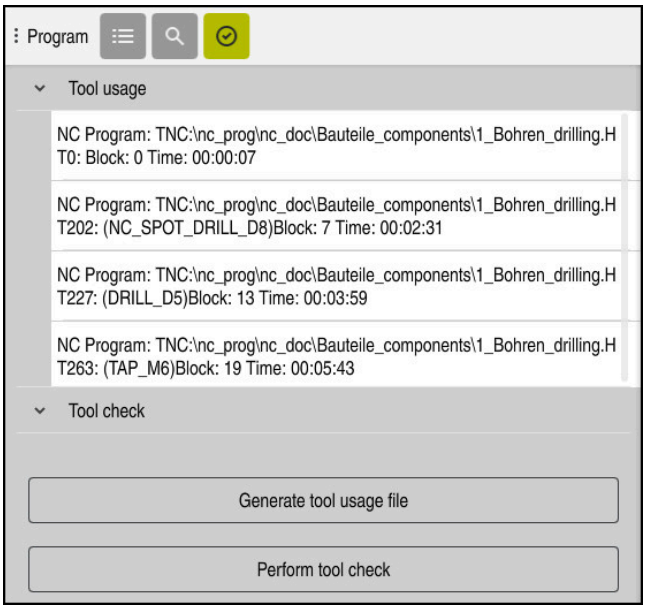
When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

- Simulating the NC program completely
- Executing the NC program completely
- Select **Generate tool usage file** in the **Tool check** column of the **Program** workspace

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 438

Tool check column in the Program workspace



Tool check column in the **Program** workspace

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

- **Tool usage**
  Further information: "Tool usage area", Page 193
- **Tool check**
  Further information: "Tool check area", Page 193

Further information: User's Manual for Programming and Testing

Tool usage area

The **Tool usage** area is empty before generating a tool usage file.

Further information: "Creating the tool usage file", Page 192

Further information: "Tool usage file", Page 438

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

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

Tool check area

Before performing a tool usage test with the **Tool check** button, the **Tool check** area has no content.

Further information: "Performing the tool usage test", Page 194

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

- The tool is defined in the tool management

  Further information: "Tool management ", Page 184
- The tool is defined in the pocket table

  Further information: "Pocket table tool_p.tch", Page 435
- The tool has sufficient remaining tool life

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

  Further information: "Tool table tool.t", Page 405

  Further information: "Tool usage file", Page 438

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

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

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

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

  Change the tool or use a replacement tool.

  Further information: User's Manual for Programming and Testing

If you double-tap or 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.

9.6.1 Performing the tool usage test

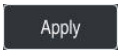
Use the tool usage test as follows:

- Select the **Home** operating mode

- Select the **Settings** application

- Select the **Machine settings** group

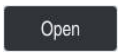
- Select the **Machine settings** menu item
- In the **Channel settings** area, select Generate tool usage file **Once** for the simulation.
  Further information: "Channel settings", Page 494



- Press **Apply**

- Select the **Editor** operating mode

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



- Select **Open**

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

- Select the **Tool check** column

> The control opens the **Tool check** column.

- Select **Generate tool usage file**

> The control generates a tool usage file and displays the tools used in the **Tool usage** area.
> **Further information:** "Tool usage file", Page 438

- Select **Perform tool check**

> The control performs the tool usage test.
> The **Tool check** area of the control shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you selected **never** in the **Generate tool-usage file** function, the **Generate tool usage file** button in the **Tool check** column is grayed out.

  Further information: "Channel settings", Page 494
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.

  Further information: User's Manual for Programming and Testing
- The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

  Further information: "Tool usage file", Page 438
- The control displays the order of tool calls of the NC program that is active in the program run in the **T usage order** table (option 93).

  Further information: "T usage order (option 93)", Page 440
- The control shows an overview of all tool calls of the NC program that are active in the program run in the **Tooling list** table (option 93).

  Further information: "Tooling list (option 93)", Page 442
- Function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- Function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. The pallet table row is defined by **IDX**.
- 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.
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files with the *.dep extension in the file manager. Even if the control displays no dependency files, it still generates a tool usage file.

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 200
B-CS	Basic coordinate system basic coordinate system	Page 202
W-CS	Workpiece coordinate system workpiece coordinate system	Page 204
WPL-CS	Working plane coordinate system working plane coordinate system	Page 206
I-CS	Input coordinate system input coordinate system	Page 209
T-CS	Tool coordinate system tool coordinate system	Page 210

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

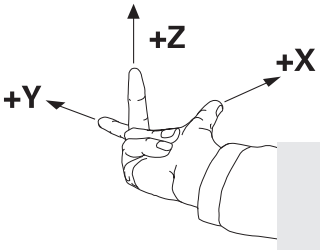
Types of coordinate systems

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.

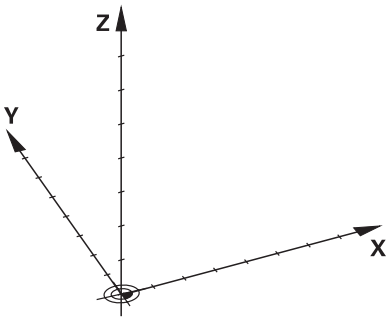
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

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

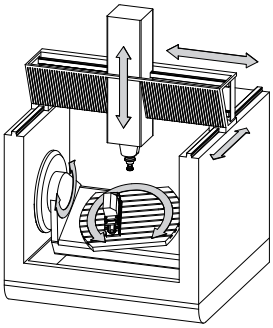
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

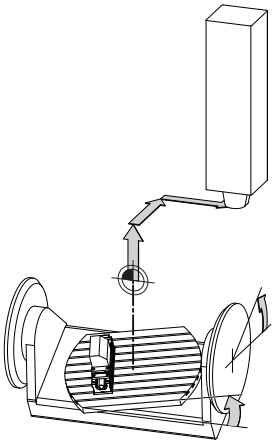
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate 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", Page 443

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

- The **Additive offset (M-CS)** function for rotary axes in the **GS** workspace (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 261

The machine manufacturer can also define further transformations.
Further information: "Note", Page 201

Position display

The following modes of the position display are referenced to the machine coordinate system **M-CS**:

- **Nominal reference position (RFNOML)**
- **Actual reference position (RFACTL)**

The difference between the values for the **RFACTL** and **ACTL.** modes of an axis result from all stated offsets as well as all active transformations in other reference systems.

Programming coordinate entry in machine coordinate system M-CS

With miscellaneous function **M91** you program the coordinates relative to the machine datum.

Further information: User's Manual for Programming and Testing

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. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets

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**.

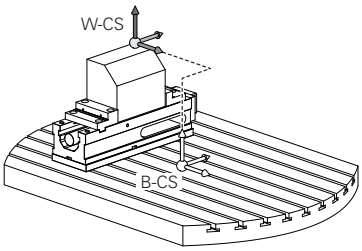
Transformations in the basic coordinate system B-CS

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 213



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

Further information: "Note", Page 203

Note

The machine manufacturer can define additional basic transformations and store them 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. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets

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.

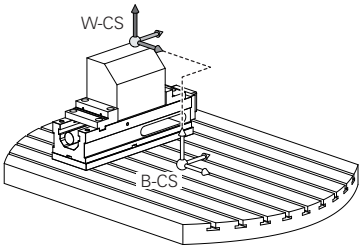
Description of function

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 213



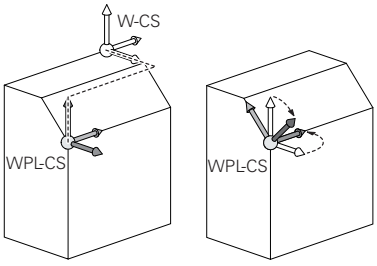
Transformations in the workpiece coordinate system (W-CS)

HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

- **TRANS DATUM** function before tilting the working plane
  Further information: User's Manual for Programming and Testing
- Function **TRANS MIRROR** or Cycle **8 MIRRORING** before tilting the working plane with spatial angles
  Further information: User's Manual for Programming and Testing
  Further information: User's Manual for Machining Cycles
- **PLANE** functions for tilting the working plane (option 8)
  Further information: User's Manual for Programming and Testing

> You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

With these transformations, the position and orientation of the working plane coordinate system **WPL-CS** are changed.



NOTICE

Danger of collision!

The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.

- Program only the recommended transformations in the respective reference system
- Use tilting functions with spatial angles instead of with axis angles
- Use the Simulation mode to test the NC program

In the machine parameter **planeOrientation** (no. 201202) the machine manufacturer defines whether the control interprets input values of Cycle **19 WORKING PLANE** as spatial angles or as axis angles.

The type of tilting function has the following effects on the result:

- If you tilt using spatial angles (**PLANE** functions except for **PLANE AXIAL** or Cycle **19**), previously programmed transformations will change the position of the workpiece datum and the orientation of the rotary axes:
  - Shifting with the **TRANS DATUM** function will change the position of the workpiece datum.
  - Mirroring changes the orientation of the rotary axes. The entire NC program, including the spatial angles, will be mirrored.
- If you tilt using axis angles (**PLANE AXIAL** or Cycle **19**), a previously programmed mirroring has no effect on the orientation of the rotary axes. You use these functions for direct positioning of the machine axes.

Additional transformations with Global Program Settings (GPS, option 44)

In the **GS** workspace (option 44) you can define 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 active in addition to a mirror image (function **TRANS MIRROR** 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 in effect after the **Shift (W-CS)** and **Mirroring (W-CS)** functions and before the working plane is tilted.

Further information: "Globale Programmeinstellungen GPS", Page

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 209
- 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 206
- 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 209

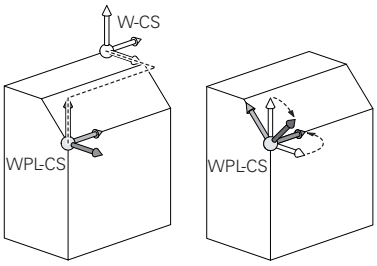
Description of function

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 204

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

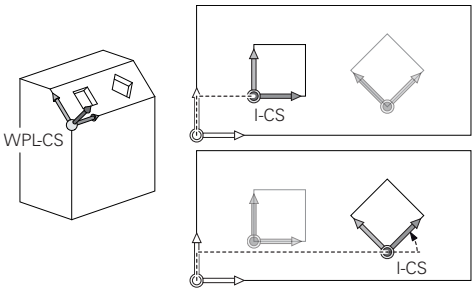


Transformations in the working plane coordinate system WPL-CS

HEIDENHAIN recommends using the following transformations in the working plane coordinate system **WPL-CS**:

- **TRANS DATUM** function
  Further information: User's Manual for Programming and Testing
- **TRANS MIRROR** or Cycle **8 MIRRORING** function
  Further information: User's Manual for Programming and Testing
  Further information: User's Manual for Machining Cycles
- **TRANS ROTATION** function or Cycle **10 ROTATION**
  Further information: User's Manual for Programming and Testing
  Further information: User's Manual for Machining Cycles
- **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
  Further information: User's Manual for Programming and Testing
  Further information: User's Manual for Machining Cycles
- Cycle **26 AXIS-SPECIFIC SCALING**
  Further information: User's Manual for Machining Cycles
- **PLANE RELATIV** function (option 8)
  Further information: User's Manual for Programming and Testing

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, option 44)

The **Rotation (WPL-CS)** transformation in the **GS** workspace has an additive effect to a rotation in the NC program.

Further information: "Global Program Settings (GPS, option 44)", Page 261

Additional transformations with mill-turning (option 50)

The following additional transformations are available with the mill-turning software option:

- 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 software option 50.
>
> 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:** "POS tab", Page 124

Additional transformation with Gear Cutting (option 157)

You can use the following cycles to define a precession angle:

- Cycle **286 GEAR HOBBING**
- Cycle **287 GEAR SKIVING**

> Machine manufacturers can also define a precession angle without Gear Cutting (software option 157)

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 209
- 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 (option 8), **PLANE RELATIV** has an effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

10.1.7 Input coordinate system I-CS

Application

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

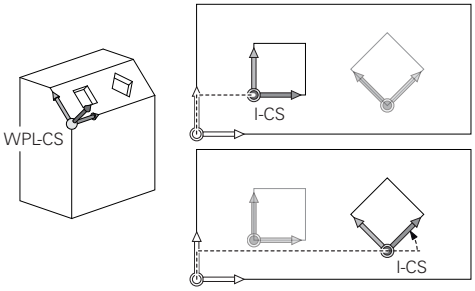
Description of function

Properties of the input coordinate system I-CS

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

Further information: "Working plane coordinate system WPL-CS", Page 206

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 210

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 (option 9)
- Cycles

11 X+48 R+	; Paraxial positioning block
11 L X+48 Y+102 Z-1.5 R0	; Path function **L**
11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0	; Straight line **LN** with Cartesian coordinates and surface normal vector

Position display

The following modes of the position display are referenced to the input coordinate system **I-CS**:

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.

  Further information: "Working plane coordinate system WPL-CS", Page 206

10.1.8 Tool coordinate system T-CS

Application

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

Description of function

Properties of the tool coordinate system T-CS

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

You make entries in the tool management to define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 154

You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool carrier reference point", Page 159

You use positioning blocks in the input coordinate system **I-CS** to define the position of the tool and therefore the position of the **T-CS**.

Further information: "Input coordinate system I-CS", Page 209

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

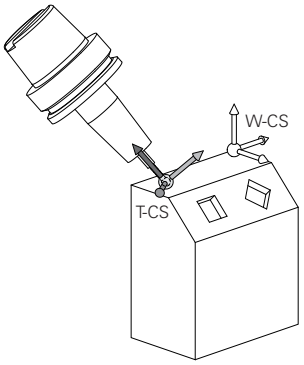
Further information: User's Manual for Programming and Testing

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

If the following functions are active, the orientation of the **T-CS** depends on the tool angle of inclination:

- Miscellaneous function **M128** (option 9)
- **PLANE RELATIV** function (option 9)

  Further information: User's Manual for Programming and Testing



Use the miscellaneous function **M128** to define the tool angle of inclination in the machine coordinate system **M-CS** using axis angles. The effects of the tool angle of inclination depend on the machine kinematics:

Further information: User's Manual for Programming and Testing

Further information: User's Manual for Programming and Testing

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 angle of inclination in the working plane coordinate system **WPL-CS**, for example with **FUNCTION TCPM** or a straight line **LN**.

11 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; **FUNCTION TCPM** with spatial angles
12 L A+0 B+45 C+0 R0 F2500	

11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 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: User's Manual for Programming and Testing
- Compensation values from the tool call
  Further information: User's Manual for Programming and Testing
- Values of the compensation tables ***.tco**
  Further information: User's Manual for Programming and Testing
- Values of **FUNCTION TURNDATA CORR T-CS** (option 50)
  Further information: User's Manual for Programming and Testing
- 3D tool compensation with surface normal vectors (option 9)
  Further information: User's Manual for Programming and Testing
- 3D tool radius compensation depending on the tool's contact angle using compensation-value tables (option 92)
  Further information: User's Manual for Programming and Testing

Position display

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 **GS workspace** (option 44) and on the **GS** tab of the **Status** workspace.

Further information: "Global Program Settings (GPS, option 44)", Page 261

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

Further information: "Contents of an electronic handwheel display", Page 462

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 the misalignment of a workpiece in the preset table. The active row of the preset table serves as the workpiece preset in the NC program and as the origin of workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 154

Use the preset management in the following cases:

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

Related topics

- Contents of preset table, write protection
  Further information: "Preset table", Page 443

Description of function

Setting presets

Presets can be set in the following ways:

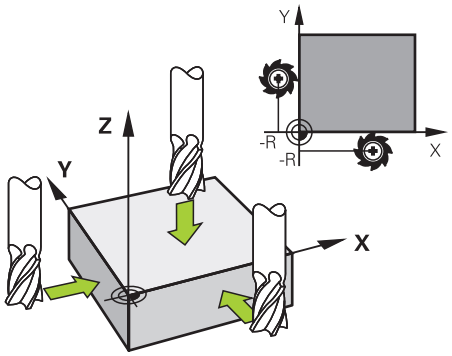
- Setting axis positions manually
  Further information: "Setting a preset manually", Page 216
- Touch probe cycles in the **Setup** application
  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 331
- Touch probe cycles in the NC program
  Further information: User's Manual for Measuring Cycles for Workpieces and Tools

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 449

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 toward the workpiece edge in the **Manual operation** application while the spindle is rotating.

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

Further information: "Setting a preset manually", Page 216

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.

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

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode
  Further information: "Activating a preset manually", Page 217
- Cycle **247 PRESETTING**
  Further information: User's Manual for Machining Cycles
- **PRESET SELECT** function
  Further information: User's Manual for Programming and Testing

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

- Datum shift with the **TRANS DATUM** function
- Mirror image 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 204

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.

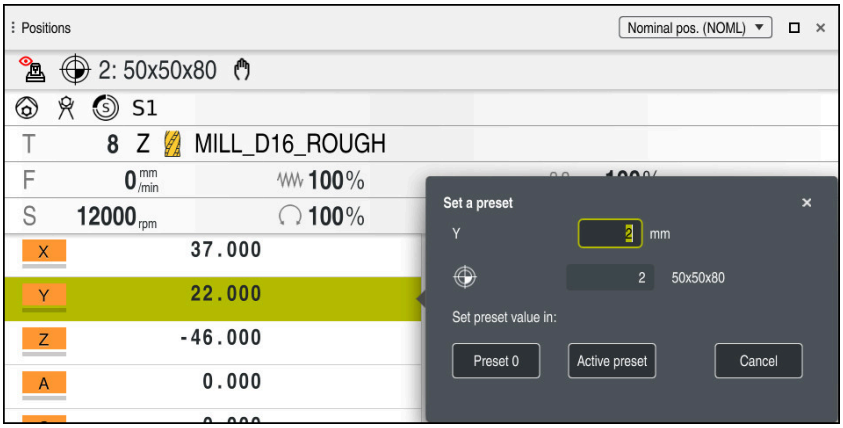
The **3D ROT** button (option 8) allows defining that the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application as well.

Further information: User's Manual for Programming and Testing

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

Further information: User's Manual for Programming and Testing

10.2.1 Setting a preset manually



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

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

To set a preset manually in an axis:

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

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

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

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

Active preset

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

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

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

- The **Set the preset** button in the function bar opens the **Set a preset** window for the row marked in green.
- When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.

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.

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

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 109

Further information: "Status overview on the TNC bar", Page 115

Notes

- The machine manufacturer uses the optional machine parameter **initial** (no. 105603) to define a default value for each 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 (option 8). 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: User's Manual for Programming and Testing

  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 363
- 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: User's Manual for Programming and Testing

10.3 Tilting the working plane (option 8)

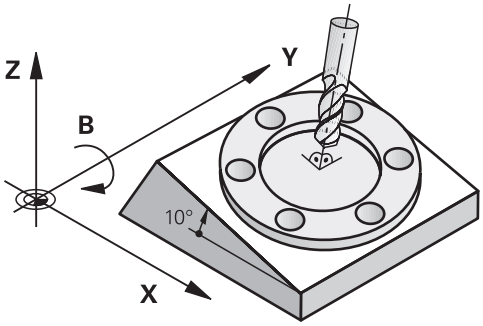
10.3.1 Fundamentals

Tilting the working plane of machines with rotary axes allows you to machine several workpiece sides in just one clamping setup, for example. 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 206



There are two functions available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application
  Further information: "3-D rotation window (option 8)", Page 220
- Tilting under program control with the **PLANE** functions in the NC program
  Further information: User's Manual for Programming and Testing

> You can still run NC programs from earlier controls that contain Cycle **19 WORKING PLANE**.

Notes concerning different machine kinematics

When no transformations are active and the working plane is not tilted, the linear machine axes move in parallel with the basic coordinate system **B-CS**. In this process, machines behave almost identically, regardless of the kinematics.

Further information: "Basic coordinate system B-CS", Page 202

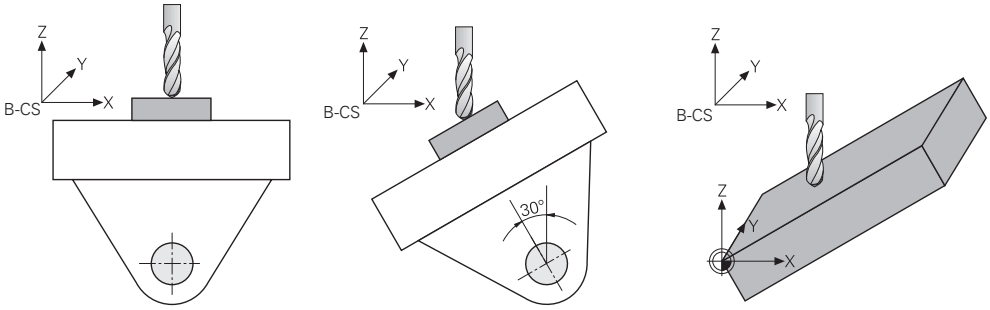
When tilting the working plane, the control moves the machine axes according to the kinematics.

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**.

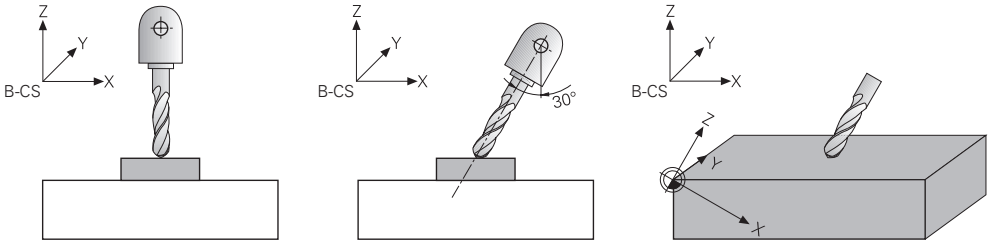
  Further information: "Working plane coordinate system WPL-CS", Page 206



- 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 206



10.3.2 3-D rotation window (option 8)

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: User's Manual for Programming and Testing
- Reference systems of the control

  Further information: "Reference systems", Page 198

Requirements

- Machine with rotary axes
- Kinematics description

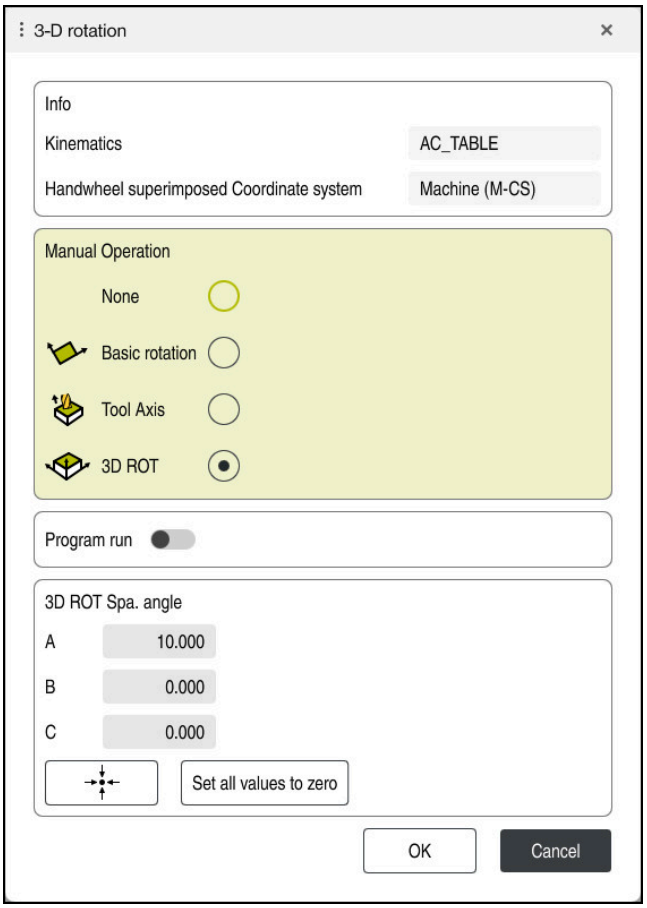
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Function enabled by the machine manufacturer

  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: "Manual operation application", Page 146



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 198 **Further information:** "Function Handwheel superimp.", Page 269 **Further information:** User's Manual for Programming and Testing

Area	Contents
Manual Operation	Effect of the tilting function in the **Manual** operating mode: ■ **None** The control will not take the rotary axes positions that are not equal to 0 into account. Traverses take place in the **W-CS** workpiece coordinate system. **Further information:** "Workpiece coordinate system W-CS", Page 204 ■ **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:** "Basic rotation selection item", Page 222 ■ **Tool axis** This is relevant only for head rotary axes. The traverses take place in the **T-CS** tool coordinate system. **Further information:** "Tool axis selection item", Page 223 ■ **3D ROT** The control takes the positions of rotary axes and columns **SPA**, **SPB** and **SPC** of the preset table into account. The traverses take place in the **WPL-CS** working plane coordinate system. **Further information:** "3D ROT selection item", Page 223
Program run	When activating the **Tilt working plane** function for the **Program run** operating mode, the entered angle of rotation applies starting from the first NC block of the NC program to be run. If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, then the angular values defined there become active. The control will reset the entered angular values to 0.
3D ROT Spa. angle	Currently active angle for the **3D ROT** selection item The machine manufacturer uses the machine parameter **planeOrientation** (no. 201202) to define whether the control calculates with spatial angles **SPA**, **SPB** and **SPC** or with the axis values of the existing rotary axes.

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

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

Further information: "Positions workspace", Page 109

Basic rotation selection item

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

Further information: "Basic rotation and 3D basic rotation", Page 215

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

Further information: "Workpiece coordinate system W-CS", Page 204

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: "Positions workspace", Page 109

The **3D ROT Spa. angle** area has no function with this selection item.

Tool axis selection item

If you select the **Tool axis** selection item, 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 210

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.

The **3D ROT Spa. angle** area has no function with this selection item.

3D ROT selection item

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

Further information: "Working plane coordinate system WPL-CS", Page 206

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

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

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

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **COORD ROT**
  - after **PLANE RESET**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- The control uses the **TABLE ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **TABLE ROT**
  - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). 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: "Referencing workspace", Page 141
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.

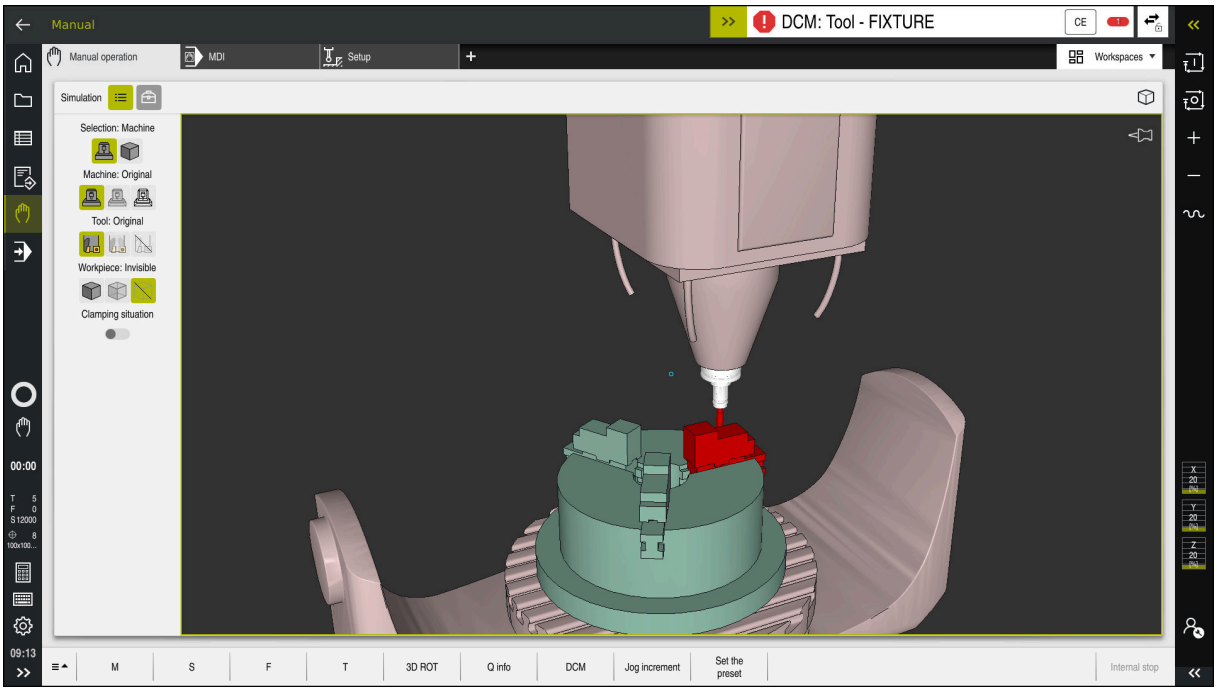
11

Collision Monitoring

11.1 Dynamic Collision Monitoring (DCM, option 40)

Application

Dynamic Collision Monitoring (DCM, dynamic collision monitoring) can be used for collision monitoring of machine components defined by the machine manufacturer. When the collision objects come closer to each other than a defined minimum distance, the control stops and displays an error message. This procedure reduces the risk of collision.



Dynamic Collision Monitoring (DCM) including collision warning

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- 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 monitoring (option 40)", Page 232
- Tools with a positive radius **R** and length **L**.

  Further information: "Tool table tool.t", Page 405
- The values in the tool management equal the actual tool dimensions

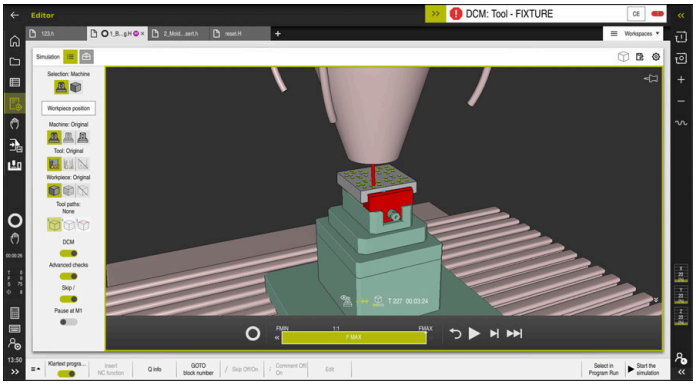
  Further information: "Tool management ", Page 184

Description of function

Refer to your machine manual.

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

The machine manufacturer can define machine components and minimum distances that are 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 inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger 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 the **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 does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

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

Further information: User's Manual for Programming and Testing

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 230

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 distance. 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: User's Manual for Programming and Testing

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

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

The enhanced collision monitoring shows collisions between the workpiece and tools or tool holders.

Further information: User's Manual for Programming and Testing

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: User's Manual for Programming and Testing

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, option 44) are not available
- Handwheel superimposition is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

11.1.1 Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger 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 the **Single Block** mode while DCM is deactivated

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

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

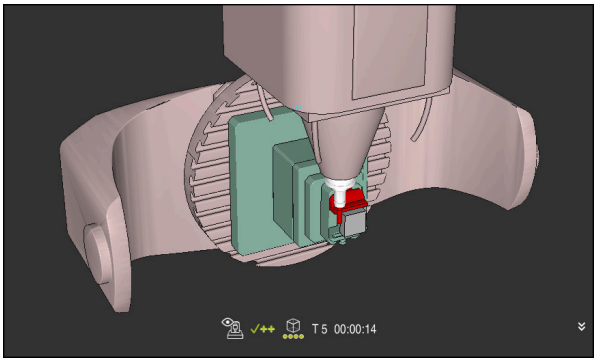
> The control opens the **Dyna. Coll. Monitoring (DCM)** window.

- 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.

11.1.2 Activating the graphic display of the collision objects



Simulation in the **Machine** mode

To activate the graphic display of the collision objects:

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

- Select **Workspaces**
- Select the **Simulation** workspace
- > The control opens the **Simulation** workspace.

- Select the **Visualization options** column
- Select the **Machine** mode
- > The control displays a graphic representation of the machine and the workpiece.

Changing the representation

To change the graphic display of the collision objects:

- Activate the graphic display of the collision objects

- Select the **Visualization options** column

- Change the graphic display of the collision objects (e.g., **Original**)

Notes

- Dynamic Collision Monitoring (DCM) helps you reduce the risk of collision. However, the control cannot consider all possible constellations during operation.
- The control can protect only those machine components from collision that your machine manufacturer has defined correctly with regard to dimensions, orientation, and position.
- The control takes the **DL** and **DR** delta values from the tool management into account. Delta values from the **TOOL CALL** block or a compensation table are not taken into account.
- For certain tools (e.g., face-milling cutters) the radius that would cause a collision can be greater than the value defined in the tool management.
- When a touch probe cycle starts, the control no longer monitors the stylus length and ball-tip diameter, so you can still probe collision objects.

11.2 Fixture monitoring (option 40)

11.2.1 Fundamentals

Application

The Fixture Monitoring function allows you to map setup situations and monitor them for collisions.

Related topics

- Dynamic Collision Monitoring (DCM, option 40)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226
- Integrating an STL file as workpiece blank
  Further information: User's Manual for Programming and Testing

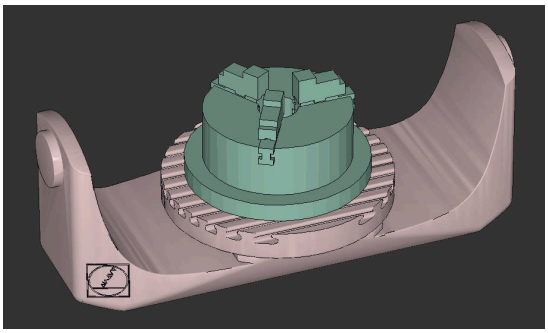
Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Kinematics description
  The machine manufacturer creates the kinematics description
- Insertion point defined
  Using the insertion point, the machine manufacturer defines the preset for positioning the fixtures. The insertion point is often located at the end of the kinematic chain (e.g., at the center of a rotary table). For information about the position of the insertion point, please refer to your machine manual.
- Fixtures of suitable format:
  - STL file
    - 20,000 triangles maximum
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

Description of function

To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading into the control
  Further information: "Options for fixture files", Page 233
- Fixture placement
  - **Set up fixtures** function in the **Setup** application (option 140)
    Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 235
  - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
  Further information: User's Manual for Programming and Testing



Three-jaw chuck loaded as fixture

Options for fixture files

For the integration of fixtures with the **Set up fixtures** function, only STL files can be used.

You can use the **3D mesh** function (option 152) 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 (option 152)", Page 318

Alternatively, CFG and M3D files can be set up manually.

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 CAD Model Optimizer (software option 152), 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 (option 152)", Page 318

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.

Fixtures from CFG files

CFG files are configuration files. You can integrate the STL and M3D files available in a CFG file. This enables you to map complex setups.

The **Set up fixtures** function can be used to create a CFG file for the fixture, using the measured value.

In CFG files, you can correct the orientation of the fixture files to be effective on the control. **KinematicsDesign** can be used to create and edit CFG files on the control.

Further information: "Editing CFG files with KinematicsDesign", Page 244

Notes

NOTICE

Danger of collision!

The setup situation defined for fixture monitoring must match the actual machine status. Otherwise, there is a risk of collision.

- Measure the position of the fixture in your machine
- Use the measured values for positioning the fixture
- Test the NC programs in the Simulation

- When using a CAM system, use a postprocessor to output the fixture situation.
- Note the orientation of the coordinate system in the CAD system. Use the CAD system to adapt the orientation of the coordinate system to the desired orientation of the fixture in the machine.
- You can choose any orientation of the fixture model in the CAD system, and therefore the orientation does not always match the orientation of the fixture in the machine.
- Define the coordinate origin in the CAD system such that the fixture can be directly attached to the point of insertion of the kinematics.
- Create a central directory for your fixtures (e.g., **TNC:\system\Fixture**).
- HEIDENHAIN recommends storing variants of recurring setup situations suitable for standard workpiece sizes in the control (e.g., vise with different jaw opening widths).

  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:

  https://www.klartext-portal.com/en/tips/nc-solutions

11.2.2 Integrating the fixtures into collision monitoring (option 140)

Application

The **Simulation** function determines the position of a 3D model in the **Set up fixtures** 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

- **Simulation** workspace
  Further information: User's Manual for Programming and Testing
- Dynamic Collision Monitoring (DCM)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226
- Fixture monitoring
  Further information: "Fixture monitoring (option 40)", Page 232
- Setting up the workpiece with graphic support (option 159)
  Further information: "Setting up the workpiece with graphical support (option 159)", Page 356

Requirements

- Dynamic Collision Monitoring (DCM version 2, software option 140)
- Workpiece touch probe
- Permitted fixture file matching the real fixture
  Further information: "Options for fixture files", Page 233

Description of function

The **Set up fixtures** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

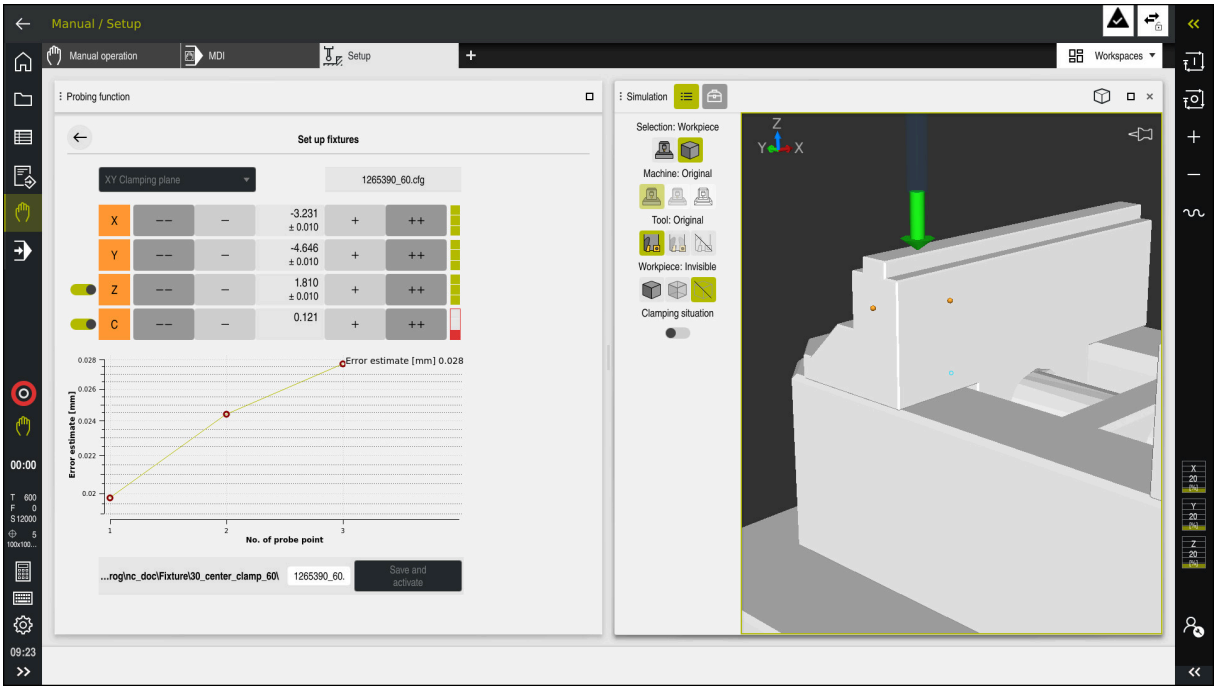
The **Set up fixtures** function determines the fixture position using various probing processes. First, one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to 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 239

Extensions of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphic support for setting up the fixture.



Set up fixtures function with open **Simulation** workspace

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	Function
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**.
127_Fixture.cfg	Name of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited before saving.
– –	Shifts the position of the virtual fixture by 10 mm or 10° in the negative axis direction Shifts the fixture in mm in a linear axis and in degrees in a rotary axis.
–	Shifts the position of the virtual fixture by 1 mm or 1° in the negative axis direction
91.349 ± 0.007	■ Enter the position of the virtual fixture directly ■ Value and estimated accuracy after probing
+	Shifts the position of the virtual fixture by 1 mm or 1° in the positive axis direction
++	Shifts the position of the virtual fixture by 10 mm or 10° in the positive axis direction
	Status of axis The control displays the following colors: ■ Gray The axis direction is deselected for this set-up process and will not be taken into account.
	■ White No probing points have been determined yet.
	■ Red The control cannot determine the fixture position in this axis direction.
	■ Yellow The position of the fixture in this axis direction already contains information. The information is not meaningful yet.
	■ Green The control can determine the fixture position in this axis direction.

Icon or button	Function
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM).

> When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by **Save and activate** at the end of the measuring process.
>
> When creating a new CFG file, enter a different file name next to the button.

When using a datum clamping system and for this reason you do not want to consider one axis direction (such as **Z**) when setting up the fixture, the axis in question can be deselected by a toggle switch. The control will not take deselected axis directions into account in the set-up process and positions the fixture by considering the remaining axis directions only.

Error estimate diagram

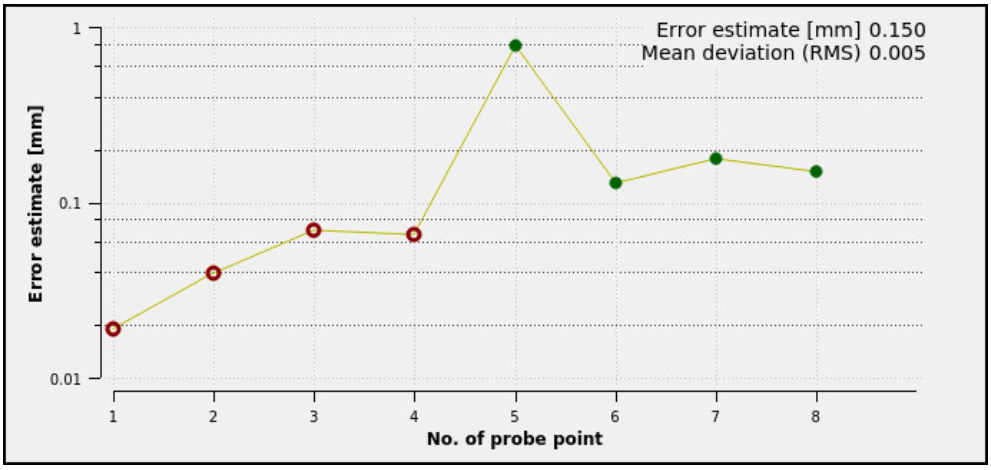
Every probing point further restricts the possible positioning of the fixture and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture. The control not only considers the probing points, but also the entire fixture.

As soon as the error estimate diagram shows green circles and the desired accuracy, the set-up process is completed.

The factors below influence the accuracy that can be achieved when measuring fixtures:

- Accuracy of workpiece touch probe
- Repeatability of workpiece touch probe
- Accuracy of 3D model
- Condition of the actual fixture (e.g., existing wear or score marks)



Error estimate diagram in the **Set up fixtures** function

The error estimate diagram of the **Set up fixtures** function displays the following information:

- **Mean deviation (RMS)**

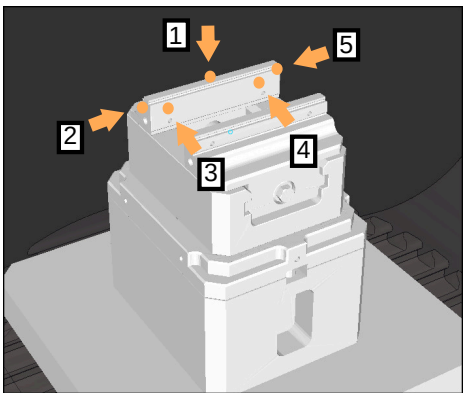
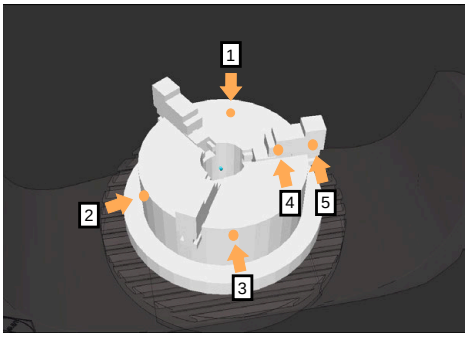
  This area shows the average distance of the measured probing points from the 3D model in mm.
- **Error estimate [mm]**

  This axis shows the course of the revised model position by means of the individual probing points. Red circles are shown until the values for all axis directions are determined. From then on, the control displays green circles.
- **No. of probe point**

  This axis shows the numbers of the individual probing points.

Example of sequence of fixture probing points

Below are some of the probing points that can be set for different fixtures:

Chucking equipment/fixtures	Possible sequence
 Probing points for a vice with a fixed vice jaw	The following probing points can be set when measuring a vice: 1 Touching the fixed vice jaw in **Z-** 2 Touching the fixed vice jaw in **X+** 3 Touching the fixed vice jaw in **Y+** 4 Touching the second value in **Y+** for rotation 5 To improve accuracy, touching the check point in **X-**
 Probing points with a three-jaw chuck	The following probing points can be set when measuring a three-point chuck: 1 Touching the jaw chuck body in **Z-** 2 Touching the jaw chuck body in **X+** 3 Touching the jaw chuck body in **Y+** 4 Touching the jaw in **Y+** for rotation 5 Touching the second value at the jaw in **Y+** for rotation

Measuring the fixed-jaw vice

> The desired 3D model must meet the requirements of the control.
> **Further information:** "Options for fixture files", Page 233

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

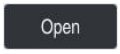
- Affix a real vice in the working space

- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the fixed vice jaw at a notable point

> This step makes the subsequent steps easier.

- Select the **Setup** application

- Select **Set up fixtures**
- > The control opens the **Set up fixtures** menu.
- Select a 3D model matching the real vice



- 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 vice, use the workpiece touch probe as a point of reference.
>
> At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using, for example, the table's T-slots produces values close to the position of the real vice.
>
> Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

- Specify the clamping plane, e. g. **XY**
- Position the workpiece touch probe until a green down arrow appears

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the fixture will actually be touched. Check if the fixture position in the simulation and in the machine match and if touching in the direction of the arrow is possible on the machine.
>
> Do not touch directly near edges, chamfers and roundings.

- Press the **NC Start** key
> The control probes in the direction of the arrow.
> The control displays the status of the **Z** axis in green and shifts the fixture to the touched position. The control marks the touched position by a point in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
> The status of the axes turns green.
- Touch another point in axis direction **Y+** for the basic rotation

> To achieve maximum accuracy when touching the basic rotation, the probing points should be as far apart from one another as possible.

> The control changes the status of the **C** axis to green.
- Touch the check point in axis direction **X-**

> Additional check points at the end of the measuring process improve the matching accuracy and minimize the faults between the 3D model and the real fixture.

Save and activate

- Select **Save and activate**
> The control closes the **Set up fixtures** function, saves a CFG file with the measured values at the path specified above, and integrates the measured fixture into Dynamic Collision Monitoring (DCM)

Notes

NOTICE

Danger of collision!

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

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

- The control cannot identify modeling differences between the 3D model and the real fixture.
- At the time of set-up, Dynamic Collision Monitoring (DCM) does not know the exact position of the fixture. In this condition, collisions with the fixture, the tool or other non-machine components such as fixing clamps in the work envelope may occur. The non-machine components can be modeled on the control using a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 244
- If you cancel the **Set up fixtures** function, DCM will not monitor the fixture. In this case, any fixtures previously set up are also removed from the scope of monitoring. The control displays a warning.
- Only one fixture can be measured at a time. To monitor several fixtures simultaneously by DCM, the fixtures must be integrated into a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 244
- When measuring a jaw chuck, the coordinates of the axes **Z**, **X** and **Y** are determined just as when measuring a vice. The rotation is determined from one single jaw.
- The saved fixture file can be integrated into the NC program with the **FIXTURE SELECT** function. This can be used for simulating and executing the NC program, considering the real setup situation.

  Further information: User's Manual for Programming and Testing

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. Several fixtures can be joined in order to take complex clamping situations into account in Dynamic Collision Monitoring (DCM).

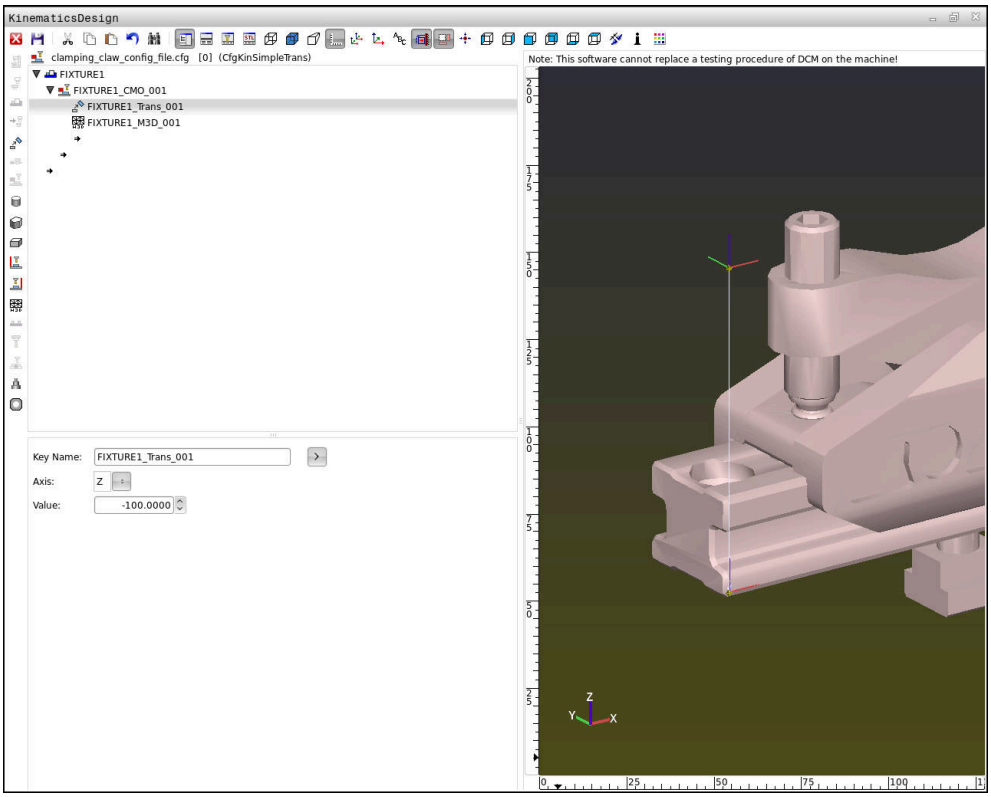
Description of function

When preparing a CFG file in the control, the control automatically opens the file with **KinematicsDesign**.

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
  - 3D model (M3D or STL files)
  - Cylinder
  - Prism
  - Cuboid
  - Truncated cone
  - Hole

You can integrate both STL files and M3D files into CFG files more than once.



Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Function	Description
`key:= ""`	Name of the function
`dir:= ""`	Direction of a transformation (e.g., **X**)
`val:= ""`	Value
`name:= ""`	Name displayed if a collision occurs (optional input)
`filename:= ""`	File name
`vertex:= [ ]`	Position of a cube
`edgeLengths:= [ ]`	Dimensions of a cuboid
`bottomCenter:= [ ]`	Center of a cylinder
`radius:=[ ]`	Radius of a cylinder
`height:= [ ]`	Height of a geometric object
`polygonX:= [ ]`	Line of a polygon in X
`polygonY:= [ ]`	Line of a polygon in Y
`origin:= [ ]`	Starting point of a polygon

Each element is assigned its own **key**. A **key** must be unambiguous and unique, meaning that it must not occur more than once in the description of a fixture. Based on the **key**, the elements are referenced to each other.

The following functions are available if you wish to use CFG functions to describe a fixture in the control:

Function	Description
`CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL",name:="")`	Definition of fixture component You can also enter an absolute path for the defined fixture component (e.g., **TNC:\nc_prog\1.STL**)
`CfgKinSimpleTrans(key:="XShiftFixture", dir:=X,val:=0)`	Shift in X axis Inserted transformations, such as a shift or rotation, are effective for all of the elements following in the kinematic chain.
`CfgKinSimpleTrans(key:="CRot0", dir:=C,val:=0)`	Rotation in C axis
`CfgCMO ( key:="fixture", primitives:= ["XShiftFixture","CRot0", "Fixture_body"], active :=TRUE, name :="")`	Describes all of the transformations contained in the fixture. The parameter active **:= TRUE** activates collision monitoring for the fixture. The **CfgCMO** contains collision objects and transformations. The fixture is combined based on the arrangement of the different transformations. Here, the transformation **XShiftFixture** shifts the center of rotation of the transformation **CRot0**.

Function	Description
`CfgKinFixModel(key:="Fix_Model", kinObjects:=["fixture"])`	Fixture designation **CfgKinFixModel** contains one or more **CfgCMO** elements.

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

The following geometric objects are available:

Function	Description
`CfgCMOCuboid ( key:="FIXTURE_Cub", vertex:= [ 0, 0, 0 ], edgeLengths:= [0, 0, 0], name:="" )`	Definition of a cuboid
`CfgCMOCylinder ( key:="FIXTURE_Cyl", dir:=Z, bottomCenter:= [0, 0, 0], radius:=0, height:=0, name:="")`	Definition of a cylinder
`CfgCMOPrism ( key:="FIXTURE_Pris_002", height:=0, polygonX:=[], polygonY:=[], name:="", origin:= [ 0, 0, 0 ] )`	Definition of a prism A prism is described by entering the height and several polygonal lines.

Creating a fixture entry with a collision object

The content below describes the procedure with **KinematicsDesign** opened.

To create a fixture entry with a collision object:

- ▶ Select **Insert chucking equipment**
- > **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a **keyname** for the fixture (e.g., **clamping jaw**)
- ▶ Confirm your input
- > **KinematicsDesign** loads the input.

- ▶ Move cursor down one level

- ▶ Select **Insert collision object**
- ▶ Confirm your input
- > **KinematicsDesign** creates a new collision object.

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:

- Create a fixture entry with a collision object

- Select the cursor key beneath the collision object

- Select the desired geometric shape (e.g., a cuboid)
- Define the position of the cuboid (e.g., **X = 0**, **Y = 0**, **Z = 0**)
- Define the dimensions of the cuboid (e.g., **X = 100**, **Y = 100**, **Z = 100**)
- Confirm your input
- > The control displays the defined cuboid in the graphic.

Integrating 3D models

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

To integrate a 3D model as a fixture:

- Create a fixture entry with a collision object

- Select the cursor key beneath the collision object

- Select **Insert 3D model**
- > The control opens the **Open file** window.
- Select the desired STL or M3D file
- Press **OK**
- > The control integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

- Define the fixture

- Select the cursor key beneath the element to be positioned

- Select **Insert transformation**
- Enter a **key name** for the transformation (e.g., **Z shift**)
- Select the **axis** for the transformation (e.g., **Z**)
- Select the **value** for the transformation (e.g., **100**)
- Confirm your input
- > **KinematicsDesign** inserts the transformation.
- > **KinematicsDesign** depicts the transformation in the graphic.

Note

As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
`CfgCMOMesh3D (key:="Fixture_body", filename:="vice_47155.STL", name:="")`	Body of the vise
`CfgCMOMesh3D (key:="vice_jaw_1", filename:="vice_jaw_47155.STL", name:="")`	First jaw of the vise
`CfgCMOMesh3D (key:="vice_jaw_2", filename:="vice_jaw_47155.STL", name:="")`	Second jaw of the vise

Definition of jaw opening width

In this example, the opening width of the vise is defined using two mutually dependent transformations.

Code	Explanation
`CfgKinSimpleTrans (key:="TRANS_opening_width", dir:=Y, val:=-60)`	Jaw opening width of the vise in Y direction: 60 mm
`CfgKinSimpleTrans (key:="TRANS_opening_width_2", dir:=Y, val:=30)`	Position of the first jaw of the vise in Y direction: 30 mm

Positioning of the fixture within the working space

The defined fixture components are positioned using various transformations.

Code	Explanation

`CfgKinSimpleTrans (key:="TRANS_X", dir:=X, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Y", dir:=Y, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z", dir:=Z, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z_vice_jaw", dir:=Z, val:=60)`
`CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180)`
`CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)`
	Positioning of the fixture components
	In this example, a rotation by 180° is inserted for rotating the defined jaw of the vise. This is necessary because the same initial model is used for both jaws of the vise.
	The rotation inserted applies to all subsequent components in the transformation chain.

Description of the fixture

You need to combine all objects and transformations in the CFG file in order to ensure that the fixture is correctly depicted in the simulation.

Code	Explanation
`CfgCMO (key:="FIXTURE", primitives:= [ "TRANS_X", "TRANS_Y", "TRANS_Z", "TRANS_SPC", "TRANS_SPB", "TRANS_SPA", "Fixture_body", "TRANS_Z_vice_jaw", "TRANS_opening_width_2", "vice_jaw_1", "TRANS_opening_width", "TRANS_C_180", "vice_jaw_2" ], active:=TRUE, name:="")`	Combining the transformations and objects contained in the fixture

Fixture designation

You need to assign a designation to the combined fixture.

Code	Explanation
`CfgKinFixModel (key:="FIXTURE1", kinObjects:=["FIXTURE"])`	Designation of the combined fixture

12

Control Functions

12.1 Adaptive Feed Control (AFC, option 45)

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 (option 45)", Page 451

Requirements

- Adaptive Feed Control (AFC, software option 45)
- 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 451
- Define settings for AFC for each tool in the tool management
  Further information: "Tool table tool.t", Page 405
- Define AFC in the NC program
  Further information: "NC functions for AFC (option 45)", Page 255
- Define AFC in the **Program Run** operating mode with the **AFC** toggle switch
  Further information: "AFC toggle switch in Program Run operating mode", Page 257
- Prior to automatic control, determine the reference spindle power with a teach-in cut
  Further information: "AFC teach-in cut", Page 258

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

Further information: "Positions workspace", Page 109

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

Further information: "AFC tab (option 45)", Page 118

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 259
- Protection of the machine's mechanical elements

  Timely feed rate reduction and shutdown responses 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 can enter 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 451
- ***.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 455
- ***.H.AFC2.DEP**

  During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you are perfoming the teach-in cut.

  In control mode, the control updates the data in this table and performs evaluations.

  Further information: "Log file AFC2.DEP", Page 456

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

Further information: "Editing tables for AFC", Page 458

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 falling below the **FMIN** value is imminent, stop the machining operation without deactivating AFC
- 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 451
  - If the programmed feed rate falls below the 30% threshold

    The control executes an NC stop.
- Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
- Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
- In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
- With the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files in the file management.

12.1.2 Activating and deactivating AFC

NC functions for AFC (option 45)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 451
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 405
- **AFC** toggle switch active

  Further information: "AFC toggle switch in Program Run operating mode", Page 257

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 the 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!

AFC toggle switch in 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 (option 45)", Page 255

Requirements

- Adaptive Feed Control (AFC, software option 45)
- 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: "Positions workspace", Page 109

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as the AFC function is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of how the function is deactivated (e.g. feed rate potentiometer). This acceleration may result in damages to the tool or the workpiece!

- If it is imminent that the feed rate falls 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%. This means that you can no longer change the spindle speed.
- If adaptive feed control is active in **Control** mode, the control loads the value from the feed-rate override function.
  - Increasing the feed-rate override has no influence on the control.
  - If you reduce the feed override with the potentiometer by more than 10% in relation to the position at the start of the program, the control switches AFC off.

    You can reactivate control with the **AFC** toggle switch.
  - Potentiometer values of up to 50% always have an effect, even with active control.
- Mid-program startup is allowed during active feed control. The control takes the cutting number of the startup block in account.

12.1.3 AFC teach-in cut

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 405
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function

  Further information: "NC functions for AFC (option 45)", Page 255

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table
  Further information: "Basic AFC settings in AFC.tab", Page 451
- Desired control setting defined for all tools
  Further information: "Tool table tool.t", Page 405
- Desired NC program selected in the **Program Run** operating mode
- **AFC** toggle switch active
  Further information: "AFC toggle switch in Program Run operating mode", Page 257

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 455

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%.

12.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear and breakage. The **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management can be used for this.

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management
  Further information: "Tool table tool.t", Page 405

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 451

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.

A 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 451

12.2 Active Chatter Control (ACC, option 145)

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 405

Requirements

- Active Chatter Control (ACC, software option 145)
- 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: "Program Run operating mode", Page 368

Further information: "The MDI Application ", Page 363

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

Further information: "Positions workspace", Page 109

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 Machine Vibration Control (MVC, software option 146), you can also positively influence the result.

12.3 Global Program Settings (GPS, option 44)

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: User's Manual for Programming and Testing
  Further information: User's Manual for Machining Cycles
- **GPS** tab in the **Status** workspace.
  Further information: "GPS tab (option 44)", Page 120
- Reference systems of the control
  Further information: "Reference systems", Page 198

Requirement

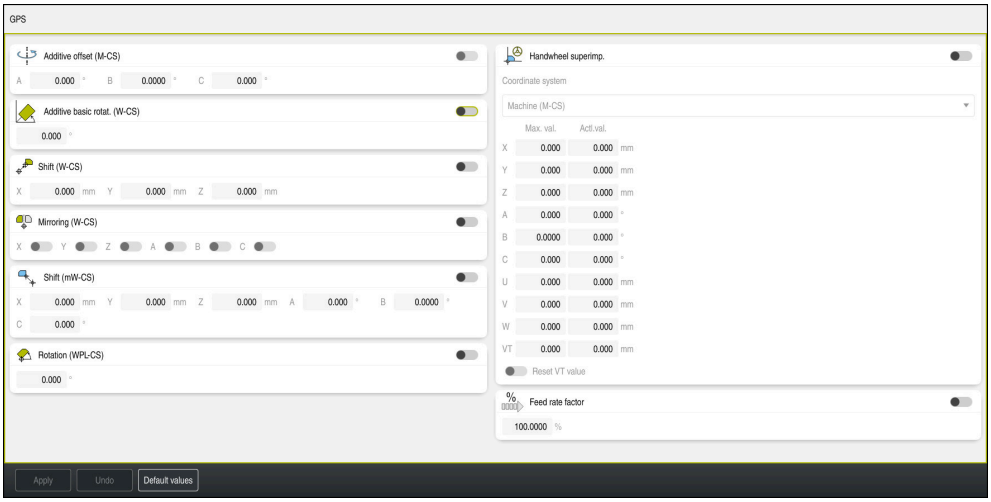
- Global Program Settings (GPS, software option 44)

Description of function

The values of the Global Program Settings are defined and activated in the **GS** workspace.

The **GS** workspace is available in the **Program Run** operating mode and in the **MDI** application of the **Manual** operating mode.

The transformations of the **GS** workspace are effective in all operating modes and are persistent across reboots of the control.



GS 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 settings of GPS on the **GPS** tab of the **Status** workspace.

Further information: "GPS tab (option 44)", Page 120

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 **GS** workspace:

Button	Description
Apply	Save changes in the **GS** workspace
Undo	Reset unsaved changes in the **GS** workspace
Default values	Set the **Feed rate factor** function to 100%, reset all other functions to zero

Overview of Global Program Settings (GPS)

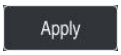
The Global Program Settings (GPS) include the following functions:

Function	Description
Additive offset (M-CS)	Shift of the zero position of an axis in the machine coordinate system **M-CS** **Further information:** "Function Additive offset (M-CS)", Page 264
Additive basic rotat. (W-CS)	Additional rotation based on basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. **Further information:** "Function Additive basic rotat. (W-CS)", Page 266
Shift (W-CS)	Shift of workpiece preset in a single axis in the workpiece coordinate system **W-CS** **Further information:** "Function Shift (W-CS)", Page 266
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system **W-CS** **Further information:** "Function Mirroring (W-CS)", Page 267
Shift (mW-CS)	Additional shift of a workpiece datum already shifted in the modified workpiece coordinate system **(mW-CS)**. **Further information:** "Function Shift (mW-CS)", Page 268
Rotation (WPL-CS)	Rotation around the active tool axis in the working plane coordinate system **WPL-CS** **Further information:** "Function Rotation (WPL-CS)", Page 269
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel **Further information:** "Function Handwheel superimp.", Page 269
Feed rate factor	Manipulation of the active feed rate **Further information:** "Function Feed rate factor", Page 272

Define and activate Global Program Settings (GPS)

To define and activate the Global Program Settings (GPS):

- Select an operating mode (e.g., **Program run**)
- Open the **GS** workspace
- Activate the toggle switch for the required 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 **GS** workspace
- Select **Default values**

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**

Apply

> The control saves the values that have been reset.

Notes

- The control grays out 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). These values include offset values and values of **Handwheel superimp.** Angles are always entered in degrees.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) 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 Function Additive offset (M-CS)

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 compensate an axis when using axis angles.

Related topics

- Machine coordinate system **M-CS**
  Further information: "Machine coordinate system M-CS", Page 200
- Difference between basic rotation and offset
  Further information: "Basic transformation and offset", Page 447

Description of function

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

Further information: "Preset table", Page 443

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: "Positions workspace", Page 109

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 **GS** workspace
- Activate the **Additive offset (M-CS)** toggle switch
- Enter **C 180°**

Apply

- 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.
- In the optional machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis how the control interprets offsets for the following NC functions:
  - **FUNCTION PARAXCOMP**
  - **FUNCTION POLARKIN** (option 8)
  - **FUNCTION TCPM** or **M128** (option 9)
  - **FACING HEAD POS** (option 50)

    Further information: User's Manual for Programming and Testing

12.3.3 Function Additive basic rotat. (W-CS)

Application

One possible use for the **Additive basic rotat. (W-CS)** function is better of the working space. 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 function **Additive basic rotat. (W-CS)** 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", Page 443

The function **Additive basic rotat. (W-CS)** 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 function **Additive basic rotat. (W-CS)**.

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 **GS** workspace
- ▶ Activate the toggle switch for **Additive basic rotat. (W-CS)**
- ▶ Enter **90°**

- ▶ Press **Apply**
- ▶ Select NC program
- > The control considers the 90° rotation for all axis positioning movements.

12.3.4 Function Shift (W-CS)

Application

You can use the **Shift (W-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult, for example.

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 204

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

Further information: "Position displays", Page 133

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 **GS** 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**
- Start an NC program
- Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- Determine the workpiece surface by scratching, using the handwheel for fine adjustment
- Select NC program

> The control takes the **Shift (W-CS)** into account.

> The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

12.3.5 Function Mirroring (W-CS)

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: User's Manual for Programming and Testing

The **Mirroring (W-CS)** function has no effect on the position display in the **Positions** workspace.

Further information: "Position displays", Page 133

Application example

An NC program can be edited with the **Mirroring (W-CS)** function mirror-inverted.

Initial situation:

- Available CAM output for right mirror cap
- NC program set to the center of the ball-nose cutter and **FUNCTION TCPM** function with spatial angles output
- The workpiece datum is centered on the workpiece blank
- Mirroring required in the X axis to produce the left mirror cap

To mirror the CAM output of an NC program:

- Open the **GS** workspace
- Activate the **Mirroring (W-CS)** toggle switch
- Activate the **X** toggle 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 **GS** 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 **GS** workspace.

12.3.6 Function Shift (mW-CS)

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 204

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

Further information: "Position displays", Page 133

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 **GS** 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 Function Rotation (WPL-CS)

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 active 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: User's Manual for Programming and Testing

The **Rotation (WPL-CS)** function has no effect on the position display.

12.3.8 Function Handwheel superimp.

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: User's Manual for Programming and Testing

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 386

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: "Positions workspace", Page 109

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: "POS HR tab", Page 125

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 219

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: "POS HR tab", Page 125

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 you want to use **Handwheel superimp.** while machining with active Dynamic Collision Monitoring (DCM), then the control must be in a stopped or interrupted state. Alternatively, you can also deactivate DCM.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 226
- **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: "Positions workspace", Page 109

12.3.9 Function Feed rate factor

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 FMAX", Page 372

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 111

13

Monitoring

13.1 Process Monitoring (option 168)

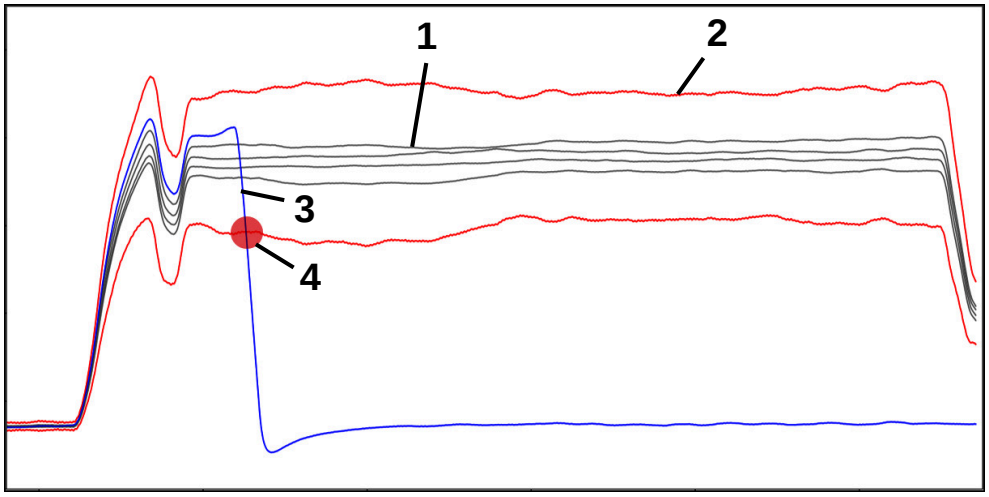
13.1.1 Fundamentals

The control uses process monitoring to detect disturbances in the machining process, e.g:.

- Tool breakage
- Incorrect or missing workpiece pre-machining
- Changed position or size of the workpiece blank
- Wrong material, e.g. aluminum instead of steel

Process monitoring allows you to monitor the machining process during program run using monitoring tasks. The monitoring task compares the signal curve of the current execution of an NC program with one or more reference machining operations. The monitoring task uses these reference machining operations to determine an upper and lower limit. If the current machining operation is outside the limits for a predefined hold time, the monitoring task executes a defined response. If, for example, the spindle current drops due to a tool breakage, the monitoring task executes a predefined response.

Further information: "Interrupting, stopping or canceling program run", Page 373



Drop in spindle current due to tool breakage

1 References
2 Limits consisting of tunnel width and, if necessary, expansion
3 Current machining operation
4 A process fault (e.g., due to tool breakage)

If you are using process monitoring, the following steps are required:

- Defining monitoring sections in the NC program
  Further information: User's Manual for Programming and Testing
- Slowly running-in the NC program in Single Block mode before activating process monitoring
  Further information: "Program Run", Page 367
- Activating process monitoring
  Further information: "Monitoring options column", Page 291
- Running the NC program in Full Sequence
  Further information: "Program Run", Page 367
- If necessary, configuring settings for the monitoring tasks
  - Selecting a strategy template
    Further information: "Strategy template", Page 281
  - Adding or removing monitoring tasks
    Further information: "Icons", Page 277
  - Defining settings and responses within the monitoring tasks
    Further information: "Monitoring task settings", Page 283
  - Displaying monitoring task in the simulation as a process heat map
    Further information: "Monitoring options column within a monitoring section", Page 292
    Further information: User's Manual for Programming and Testing
- Running the NC program again in Full Sequence operating mode
  Further information: "Program Run", Page 367
- Selecting other references and optimizing parameters
  Further information: "Monitoring tasks", Page 282
  Further information: "Records of monitoring sections", Page 294

Related topics

- **Component monitoring** (option 155) with **MONITORING HEATMAP**
  Further information: User's Manual for Programming and Testing

13.1.2 Process Monitoring workspace (option 168)

Application

In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.

Further information: "Monitoring tasks", Page 282

Requirements

- Process Monitoring (software option 168)
- Monitoring sections have been defined with **MONITORING SECTION**

  Further information: User's Manual for Programming and Testing
- Reproducibility of the process in the **FUNCTION MODE MILL** machining mode

  The monitoring tasks **FeedOverride** and **SpindleOverride** are functional in the **FUNCTION MODE TURN** machining mode (option 50).

  Further information: User's Manual for Programming and Testing

Functionality

The **Process Monitoring** workspace provides information and settings for monitoring the machining process.

Depending on the cursor position in the NC program, the control provides the following areas:

- Global area

  The control shows information about the active NC program.

  Further information: "Global area", Page 278
- Strategy area

  The control shows the monitoring tasks and the graphs of the recordings. You can configure settings for the monitoring tasks.

  Further information: "Strategy area", Page 280
- **Monitoring options** column in the global area

  The control displays information on the recordings that relate to all monitoring sections of the NC program.

  Further information: "Monitoring options column in the global area", Page 292
- **Monitoring options** column within a monitoring section

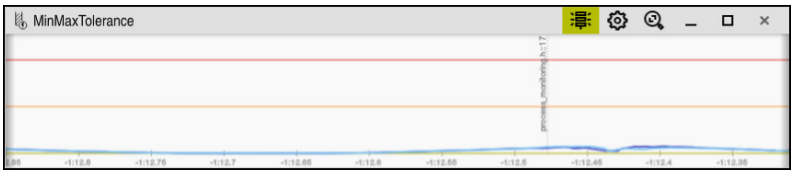
  The control displays information on the recordings that relate only to the currently selected monitoring section.

  Further information: "Monitoring options column within a monitoring section", Page 292

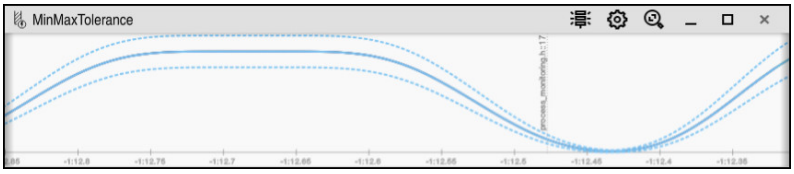
Icons

The following icons are shown in the **Process Monitoring** workspace:

Icon	Meaning
	Show or hide the **Monitoring options** column **Further information:** "Monitoring options column", Page 291
	Switch on/off setup mode If setup mode is active, the control displays the settings for process monitoring. For running the program, you can switch off setup mode.
	Remove monitoring task **Further information:** "Monitoring tasks", Page 282
	Add monitoring task **Further information:** "Monitoring tasks", Page 282
	Open settings You can open the following settings: ■ **Process Monitoring** workspace setting **Further information:** "Settings for the Process Monitoring workspace", Page 290 ■ Setting in the **Settings for NC program** window of the **Monitoring options** column **Further information:** "Settings for NC program window", Page 297 ■ Monitoring task setting **Further information:** "Monitoring task settings", Page 283
	Set the graph size to 100%
	Show or hide warning and error limits If you show the warning and error limits, the control shows the monitored signal in relation to the defined limits. The control shows the following warning and error limits: ■ Green line If the current machining operation is at the bottom line, the current machining operation corresponds to the reference. ■ Orange line This line shows the warning limit. If the current machining operation exceeds the middle line, the current machining operation deviates by half the set limit of the reference. ■ Red line This line shows the error limit. If the current machining operation exceeds the upper line for a defined hold time, the monitoring task triggers a defined response (e.g., NC stop). If you hide the warning and error limits, the control shows an absolute display of the monitored signal. The dashed lines represent the upper and lower error limits, i.e. the tunnel width.



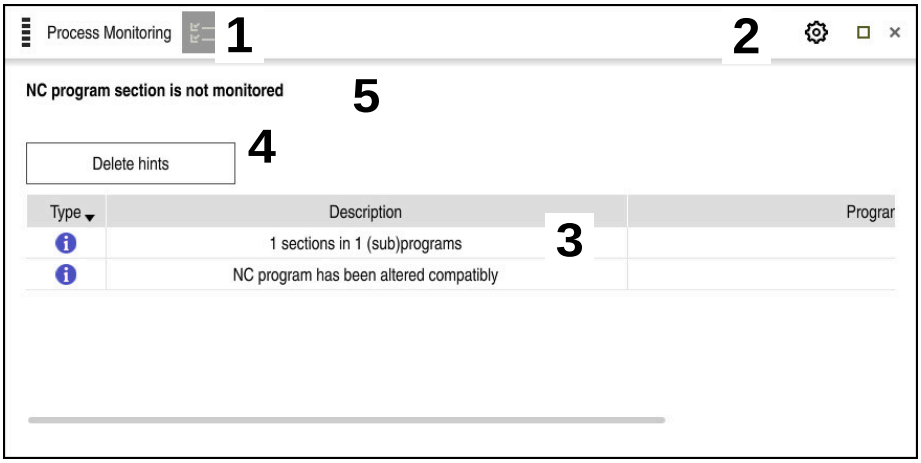
Warning and error limits displayed: The control shows the signal in relation to the defined limits



Warning and error limits hidden: The solid line represents the signal and the dashed lines represent the tunnel width determined at the time

Global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the global area.



Global area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the global area:

1 **Monitoring options** icon
Further information: "Monitoring options column", Page 291

2 **Settings** icon for the **Process Monitoring** workspace
Further information: "Settings for the Process Monitoring workspace", Page 290

3 Table with notes on the active NC program
Further information: "Notes on the NC program", Page 279

4 **Delete hints** button
You can use the **Delete hints** button to empty the table.

5 Information that this area is not monitored in the NC program

Notes on the NC program

In this area, the control shows a table with information about the active NC program. The table contains the following information:

Column or symbol	Meaning
Type	In the **Type** column, the control shows different types of notifications.
	Information (for example, the number of monitoring sections)
	Warning (for example, whether a monitoring section has been removed)
	Error (for example, whether you should reset the recordings) If you make changes within a monitoring section, that monitoring section can no longer be monitored. Therefore, you should reset the recordings and set new references so that machining is monitored again. **Further information:** "Settings for NC program window", Page 297 You can sort the table by information type by selecting the **Type** column.
Description	In the **Description** column, the control displays information about the information types, e.g.: ■ Changes to the NC program ■ Cycles contained in the NC program ■ Interruptions (e.g., **M0** or **M1**)
Program line	If the information depends on an NC block number, the control displays the program name and the NC block number.

Strategy area

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the strategy area.

Strategy area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the strategy area:

1 **Monitoring options** icon
Further information: "Monitoring options column", Page 291

2 Switch on/off setup mode
Further information: "Icons", Page 277

3 **Settings** icon for the **Process Monitoring** workspace
Further information: "Settings for the Process Monitoring workspace", Page 290

4 **Settings** icon for the monitoring tasks
Further information: "Monitoring task settings", Page 283

5 Show or hide warning and error limits
Further information: "Icons", Page 277

6 Monitoring tasks
Further information: "Monitoring tasks", Page 282

7 The control shows the following information and functions:
- Name of the monitoring section, if applicable
  If **AS** is defined in the NC program with the optional syntax element, the control displays the name.
  If no name is defined, the control displays **MONITORING SECTION**.
  Further information: User's Manual for Programming and Testing
- Range of NC block numbers of the monitoring section in square brackets
  Start and end of the monitoring section in the NC program
- **Original strategy** or **Save strategy as template** button
  Further information: "Strategy template", Page 281
- Selection menu for strategy template
  Further information: "Strategy template", Page 281

Strategy template

A strategy template includes one or more monitoring tasks, including the defined settings.

You can choose between the following strategy templates using a selection menu:

Strategy template	Meaning
MinMaxTolerance	This strategy template includes the following monitoring tasks: ■ **MinMaxTolerance** **Further information:** "Monitoring task MinMaxTolerance", Page 284 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 288 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 288 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 289
StandardDeviation	This strategy template includes the following monitoring tasks: ■ **StandardDeviation** **Further information:** "Monitoring task StandardDeviation", Page 287 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 288 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 288 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 289
User-defined	In this strategy template, you can compile the monitoring tasks yourself.

If you modify a strategy template, you can overwrite the modified strategy template by clicking the **Save strategy as template** button. The control overwrites the currently selected strategy template.

> Since you cannot restore the as-delivered state of the strategy templates yourself, only overwrite the **User-defined** template.
>
> The machine manufacturer can use the optional machine parameter **ProcessMonitoring** (no. 133700) to restore the as-delivered state of the strategy templates.

In the settings of the **Process Monitoring** workspace, you can define which strategy template the control selects by default after creating a new monitoring section.

Further information: "Settings for the Process Monitoring workspace", Page 290

Monitoring tasks

The **Process Monitoring** workspace contains the following monitoring tasks:

- **MinMaxTolerance**

  With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references, including the predefined percentage and static deviation.

  Further information: "Monitoring task MinMaxTolerance", Page 284
- **StandardDeviation**

  With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including static expansion and a multiple of the standard deviation σ.

  Further information: "Monitoring task StandardDeviation", Page 287
- **SignalDisplay**

  With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

  Further information: "Monitoring task SignalDisplay", Page 288
- **SpindleOverride**

  With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

  Further information: "Monitoring task SpindleOverride", Page 288
- **FeedOverride**

  With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

  Further information: "Monitoring task FeedOverride", Page 289

In each monitoring task, the control shows the current processing and the selected references as a graph. The time axis is specified in seconds, or in minutes for longer monitoring sections.

Monitoring task settings

You can change the settings of the monitoring tasks for each monitoring section. When you select the settings of a monitoring task, the control displays two areas. In the area on the left, the settings that were active at the time of the selected recording are grayed out. In the area on the right, the current monitoring task settings are shown. The **Apply** button allows you to save the settings from the area on the left or right. You can also remove a monitoring task from a monitoring section or add one using the plus sign.

The default values of the monitoring tasks are recommended initial values. These initial values can be adjusted for your machining purposes.

If you change the settings of a monitoring task or add a new monitoring task, the changes are identified by the ***** character preceding the name.

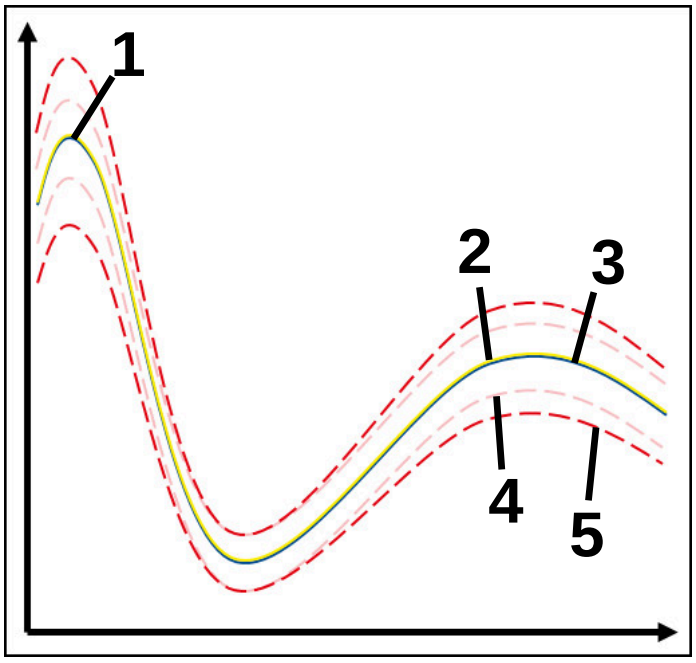
Monitoring task MinMaxTolerance

With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references, including the predefined percentage and static deviation.

The application cases of **MinMaxTolerance** are significant process faults (e.g., during small series production):

- Tool breakage
- Missing tool
- Changed position or size of the workpiece blank

The control requires at least one recorded machining operation as a reference. If you do not select a reference, this monitoring task is inactive and does not draw a graph.



1 First good reference
2 Second good reference
3 Third good reference
4 Limits consisting of the tunnel width
5 Limits consisting of a percentage expansion of the static tunnel width

Further information: "Records of monitoring sections", Page 294

If, for example, you have a recording that is only just acceptable due to tool wear, you can also use an alternative application with this monitoring task.

Further information: "Alternative application with acceptable reference", Page 286

Settings for MinMaxTolerance

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Percentage expansion of tunnel width
- **Static tunnel width**

  Upper and lower limits, based on references
- **Hold time**

  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**

  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.

  Further information: "Message menu on the information bar", Page 328
- **Trigger NC stop**

  If the signal exceeds the limits for more than the defined hold time, 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.
- **Abort program run**

  If the signal exceeds the limits for more than the defined hold time, the control aborts the NC program. In this case, the NC program cannot be resumed.
- **Lock the tool**

  If the signal exceeds the warning limits for more than the defined hold time, the control locks the tool in tool management.

  Further information: "Tool management ", Page 184

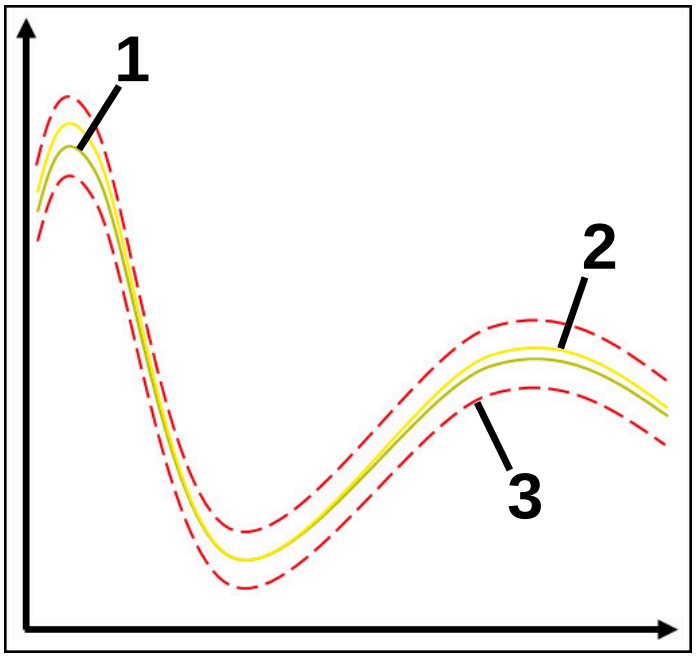
Alternative application with acceptable reference

If the control has recorded a machining operation that is only just acceptable, you can use an alternative application of the monitoring task **MinMaxTolerance**.

Select at least two references:

- An optimal reference
- A reference that is only just acceptable, e.g. showing a higher signal of the spindle load due to tool wear

The monitoring task checks whether the current machining operation is within the range of the selected references. For this strategy, select no deviation or a low percentage deviation, since the tolerance is already given by the different references.



1 Optimal reference
2 Reference only just acceptable
3 Limits consisting of the tunnel width

Monitoring task StandardDeviation

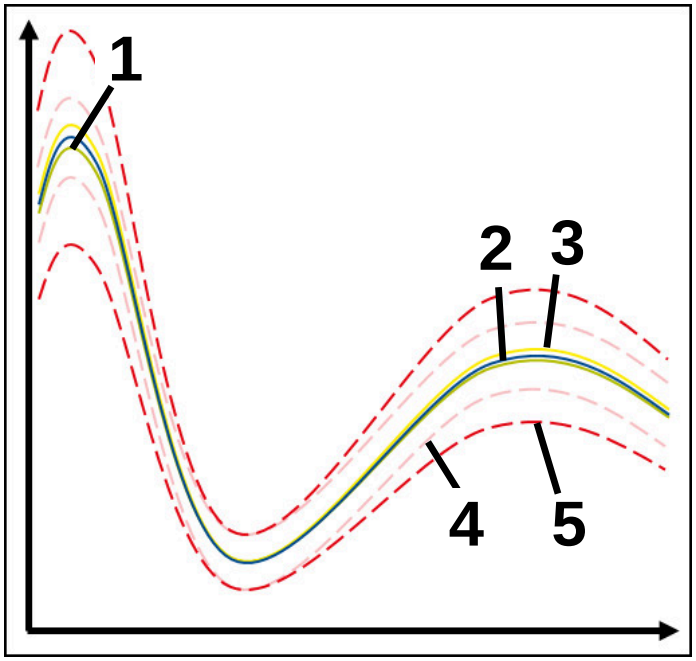
With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including static expansion and a multiple of the standard deviation σ.

The application cases of **StandardDeviation** are process faults of all kinds (e.g., during series production):

- Tool breakage
- Missing tool
- Tool wear
- Changed position or size of the workpiece blank

The control requires at least three recorded machining operations as references. The references should include one optimal, one good and one only just acceptable machining operation. If you do not select the required references, this monitoring task is not active and does not draw a graph.

Further information: "Records of monitoring sections", Page 294



1 Optimal reference
2 Good reference
3 Reference only just acceptable
4 Limits consisting of the tunnel width
5 Limits consisting of the expansion of the tunnel width multiplied by factor σ

Settings for StandardDeviation

You can use sliders to configure the following settings for this monitoring task:

- **A multiple of σ**

  Expansion of the tunnel width multiplied by factor σ
- **Static tunnel width**

  Upper and lower limits, based on references
- **Hold time**

  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**

  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.

  Further information: "Message menu on the information bar", Page 328
- **Trigger NC stop**

  If the signal exceeds the limits for more than the defined hold time, 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.
- **Abort program run**

  If the signal exceeds the limits for more than the defined hold time, the control aborts the NC program. In this case, the NC program cannot be resumed.
- **Lock the tool**

  If the signal exceeds the warning limits for more than the defined hold time, the control locks the tool in tool management.

  Further information: "Tool management ", Page 184

Monitoring task SignalDisplay

With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

You can compare whether the current machining operation corresponds to the references. This allows you to visually check whether you can use the machining operation as a reference.

The monitoring task does not respond.

Monitoring task SpindleOverride

With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

The control uses the first recorded machining operation as a reference.

Settings for SpindleOverride

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Accepted deviation of the override in percent compared to the first recording
- **Hold time**

  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Trigger warning**

  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.

  Further information: "Message menu on the information bar", Page 328
- **Trigger NC stop**

  If the signal exceeds the limits for more than the defined hold time, 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.

Monitoring task FeedOverride

With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

The control uses the first recorded machining operation as a reference.

FeedOverride settings

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Accepted deviation of the override in percent compared to the first recording
- **Hold time**

  Maximum period of time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control will trigger the responses defined for the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

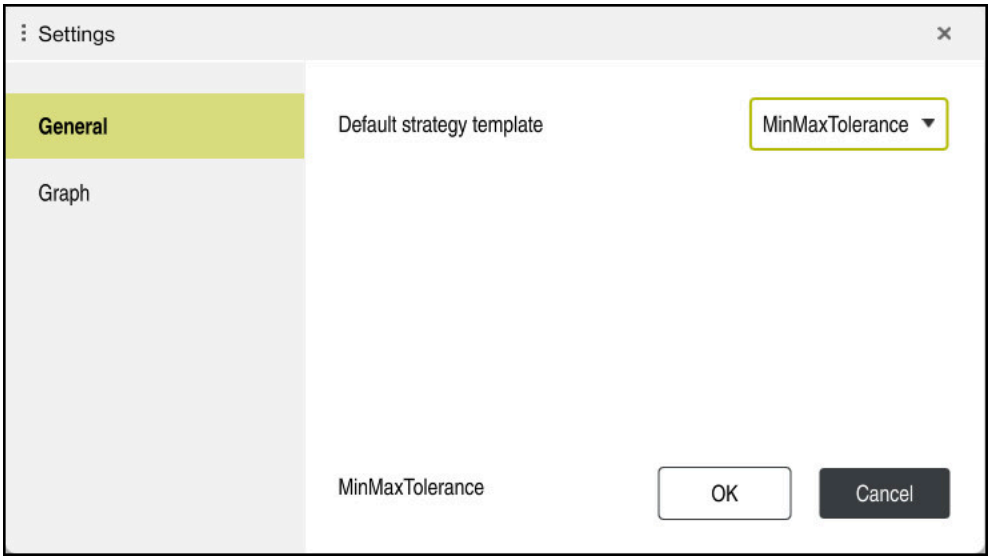
- **Trigger warning**

  If the signal exceeds the limits for more than the defined hold time, the control shows a warning in the message menu.

  Further information: "Message menu on the information bar", Page 328
- **Trigger NC stop**

  If the signal exceeds the limits for more than the defined hold time, 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.

Settings for the Process Monitoring workspace



Settings for the **Process Monitoring** workspace

General

In the **General** area, select which strategy template the control uses as the default:

- **MinMaxTolerance**
- **StandardDeviation**
- **User-defined**

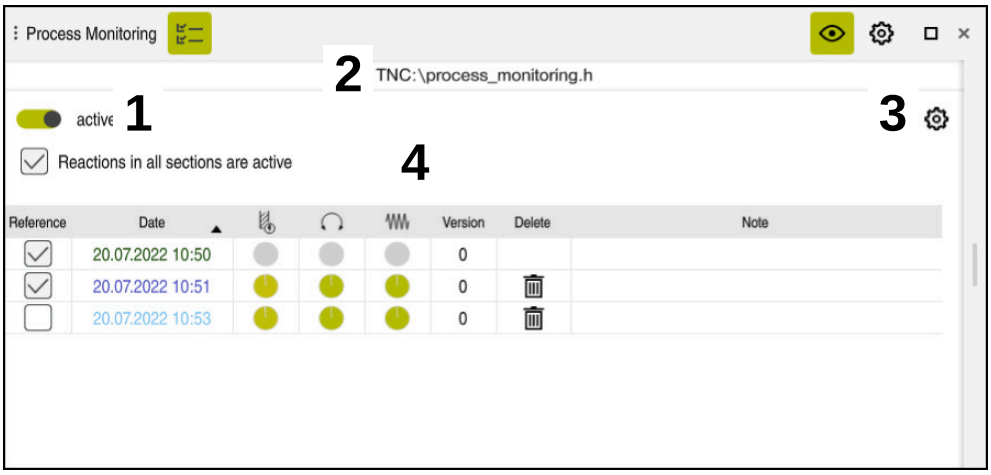
Further information: "Strategy template", Page 281

Graph

In the **Graph** area, you can select the following settings:

Setting	Meaning
Simultaneously plotted references	Select the maximum number of recordings that the control displays simultaneously as graphs in the monitoring tasks: ■ **2** ■ **4** ■ **6** ■ **8** ■ **10** If more references are selected than the control is to display, the last selected references will be displayed as recordings.
Preview [s]	The control can run selected references as a preview during program execution. The control then shifts the time axis of the machining operation to the left. Select how many seconds of the reference the control will preview: ■ **0** ■ **2** ■ **4** ■ **6** **Further information:** "Records of monitoring sections", Page 294

Monitoring options column



Monitoring options column in the global range

The **Monitoring options** column shows the following in the upper area regardless of the cursor position in the NC program:

1 Toggle switch for activating or deactivating process monitoring for the entire NC program

2 Path of the current NC program

3 Open the **Settings** icon in the **Settings for NC program** window

Further information: "Settings for NC program window", Page 297

4 Checkbox for activating or deactivating the responses of all monitoring sections in the NC program

Depending on the cursor position in the NC program, the control provides the following areas:

- **Monitoring options** column in the global area

  You can select references that are effective for all monitoring sections of the NC program.

  Further information: "Monitoring options column in the global area", Page 292
- **Monitoring options** column within a monitoring section

  You can define settings and select references that apply to the currently selected monitoring section.

  Further information: "Monitoring options column within a monitoring section", Page 292

Monitoring options column in the global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column in the global area.

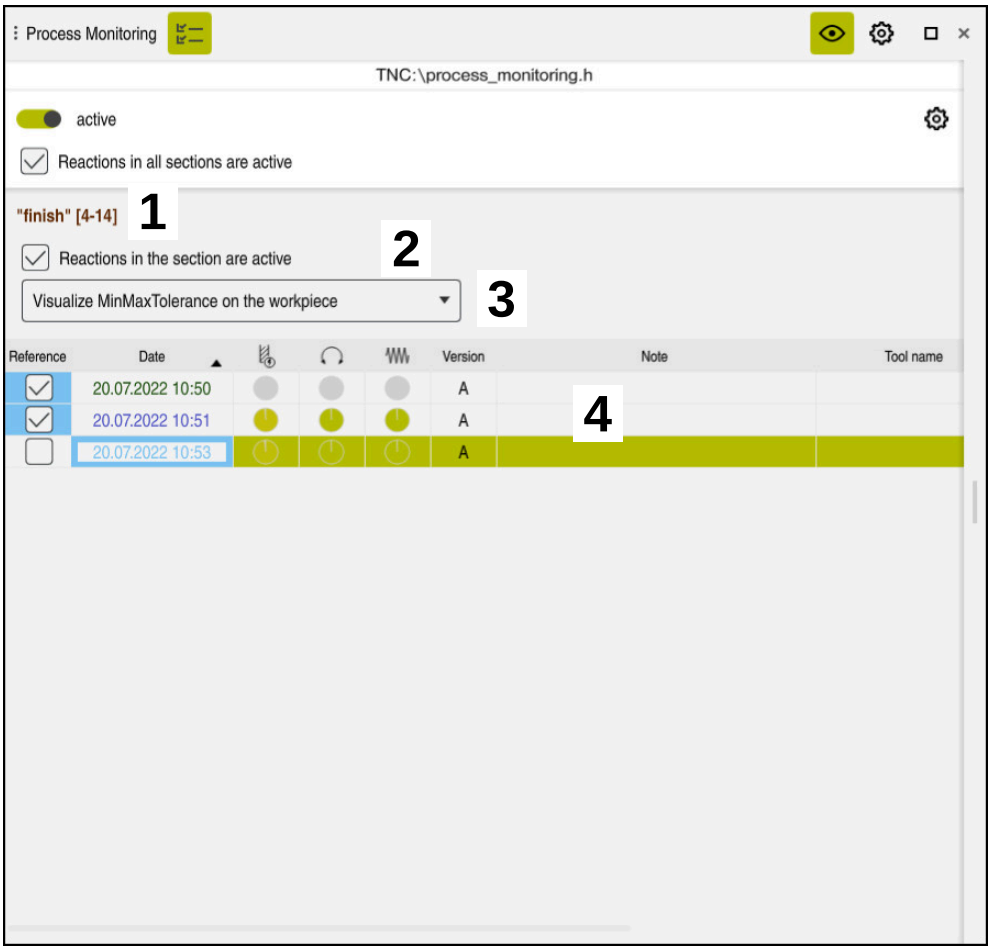
In the global area, the control displays a table with the records of all monitoring sections of the NC program.

Further information: "Records of monitoring sections", Page 294

Monitoring options column within a monitoring section

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column within the monitoring section.

If the cursor is within the monitoring section, the control will leave this area gray.



Monitoring options column within the monitoring section

The **Monitoring options** column displays the following within the monitoring section:

1 The control shows the following information and functions:
- Name of the monitoring section, if applicable

  If **AS** is defined in the NC program with the optional syntax element, the control displays the name.

  If no name is defined, the control displays **MONITORING SECTION**.

  Further information: User's Manual for Programming and Testing
- Range of NC block numbers of the monitoring section in square brackets

  Start and end of the monitoring section in the NC program

2 Checkbox for activating and deactivating the responses in the monitoring section

  You can activate or deactivate the responses of the currently selected monitoring section.

3 Selection menu for the process heat map

  You can display a monitoring task as a process heat map in the **Simulation** workspace.

  Further information: User's Manual for Programming and Testing

4 Table with the recordings of the monitoring section

  The recordings refer only to the monitoring section in which the cursor is currently located.

  Further information: "Records of monitoring sections", Page 294

Records of monitoring sections

The contents and functions of the table with the recordings of the machining operations depend on the cursor position in the NC program.

Further information: "Monitoring options column", Page 291

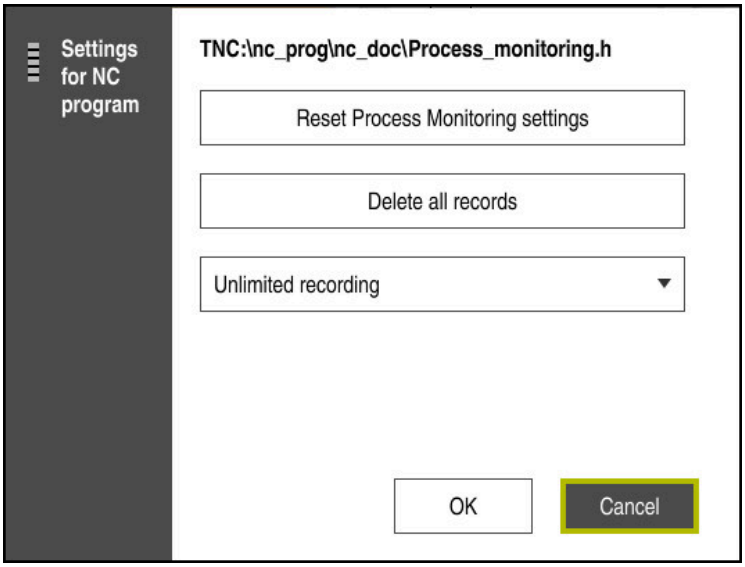
The table contains the following information about the monitoring section:

Column	Information or action
Reference	If you activate the checkbox for a table row, the control uses this recording as a reference for the corresponding monitoring tasks. If you activate multiple table rows, the control uses all selected rows as references. If you select multiple references with greater deviation, the tunnel width also increases. You can select up to ten references at a time. The effect of the reference depends on the position of the cursor in the NC program: ■ Within the monitoring section: The reference only applies to the currently selected monitoring section. The control shows a dash in the global range in this table row for information. If a table row is marked as a reference in all strategy areas or in the global area, the control displays a check mark. ■ Global area: The reference applies to all monitoring sections of the NC program. Mark recordings that have a provided satisfactory result, such as a clean surface, as a reference. You can only select a recording of a complete machining operation as a reference. When you select a recording, the control color-highlights the references selected for this recording in this column.
Date	The control displays the date and time of the program start or the starting time of the monitoring section of each recorded machining operation. If you select the **Date** column, the control sorts the table by date.

Column	Information or action
   	The control displays a color representation of the coverage of each monitoring task. The coverage indicates the percentage to which the graph of the recording corresponds to the reference graph. Warning and error limits are displayed in color. When you select a row in this column, the control displays the coverage as a percentage. If setup mode is active, the control displays the coverage as a pie chart. If the coverage is at about 80%, the machining operation is still OK. In case of lower values, make sure to check the machining process. Coverage depends on the following factors: ■ Time delay (e.g., change of the feed-rate override) If the potentiometer position of the feed-rate override deviates from the reference machining operation, the coverage becomes worse. ■ Local delay (e.g., due to tool compensation with **DR**) If the path of the tool center point (**TCP**) deviates from the reference machining operation, the coverage becomes worse. **Further information:** "Tool center point (TCP, tool center point)", Page 161 In this column, the control displays notes on the responses defined for the monitoring tasks. When you select a table cell that contains a note, the control displays detailed information on the response.
Version	If you defined custom process-monitoring settings, the control displays a different version in this column. In the **Version** column, the control displays the following information, depending on the area: ■ Within the monitoring section: The control displays letters for the different versions within the monitoring section. ■ Global area: The control displays numbers for the different versions within at least one monitoring section.
Delete	If you select the trash bin icon, the control deletes the table row and the associated recorded process data. You cannot delete the first row of the table because this row is used as a reference for the following functions: ■ For the Quality column ■ **SpindleOverride** monitoring task ■ **FeedOverride** monitoring task You delete all recordings including the first in the **Settings for NC program** window. Only in the global area
Note	In the **Note** column, you can enter notes about the table row.
Tool name	Name of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 184

Column	Information or action
R	Radius of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 184
DR	Delta value of the tool radius from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 184
L	Length of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 184
CUT	Number of cutting edges of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 184
CURR_TIME	Tool life from the tool management at the start of the respective machining operation Only within the monitoring section **Further information:** "Tool management ", Page 184

Settings for NC program window



Settings for NC program window

The **Settings for NC program** window provides the following settings:

- **Reset Process Monitoring settings**
- **Delete all records**, including the first table row
- Selection menu that displays the type and number of recorded machining operations
  - **Standard recording**

    The control records all information.
  - **Limit recordings**

    The control records all machining operations up to a certain count.

    If the number of machining operations exceeds the maximum number, the control overwrites the last machining operation.

    Input: **2...999999999**
  - **Only meta-information**

    The control does not record any process data, but only meta-information, such as the date and time. This means that you cannot use this recording as a reference. This setting can be used for monitoring and logging once process monitoring has been set up completely. This setting significantly reduces the amount of data.
  - **Each nth recording**

    The control does not record process data for each machining operation. You can define after which number of machining operations the control records process data. For the other machining operations, only meta-information will be recorded.

    Input: **2...20**

Further information: "Records of monitoring sections", Page 294

Notes

- If you use different sizes of workpiece blanks, set process monitoring to a more tolerant setting or start the first monitoring section after pre-machining.
- If the spindle load is too low, the control may not detect any difference from idling (e.g., for a tool with a small diameter).
- If you remove and add a monitoring task again, the previous recordings remain.
- 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).

Basics of operation

- You can zoom in or out of the graph horizontally by dragging or scrolling.
- If you drag or swipe with the left mouse button held down, you can move the graph.
- You can align the graph by selecting an NC block number. The control marks the selected NC block number in green within the monitoring task.
- If you double-tap or double-click a position within the graph, the control selects the corresponding NC block in the program.

  Further information: "Common gestures for the touchscreen", Page 85

13.1.3 Defining monitoring sections with MONITORING SECTION (option 168)

Application

With the **MONITORING SECTION** function, you divide the NC program into monitoring sections for process monitoring.

Related topics

- **Process Monitoring** workspace

  Further information: "Process Monitoring workspace (option 168)", Page 276

Requirement

- Process Monitoring (software option 168)

Description of function

MONITORING SECTION START is used to define the start of a new monitoring section and **MONITORING SECTION STOP**, to define the end of the monitoring section.

You must not nest monitoring sections.

If you do not define a **MONITORING SECTION STOP**, the control still interprets a new monitoring section for the following functions:

- For a new **MONITORING SECTION START**
- For a physical **TOOL CALL**

  The control only interprets a new monitoring section for a tool call when a tool change takes place.

  Further information: User's Manual for Programming and Testing

If you program the following syntax elements, the control displays a note:

- Positions relative to the machine datum (e.g., **M91**)
- Call of a replacement tool with **M101**
- Automatic liftoff with **M140**
- Repeats with variable values (e.g., **CALL LBL 99 REP QR1**)
- Jump commands (e.g., **FN 5**)
- Spindle-related M functions (e.g., **M3**)
- New monitoring section defined by **TOOL CALL**
- Monitoring section ended by **PGM END**

Further information: "Notes on the NC program", Page 279

If you program the following syntax elements, the control displays an error:

- Syntax error within a monitoring section
- Stop within the monitoring section (e.g., **M0**)
- Call of an NC program within the monitoring section (e.g., **PGM CALL**)
- Missing subprograms
- End of a monitoring section precedes the start of that monitoring section
- Multiple monitoring sections with the same contents

If an error is displayed, process monitoring cannot be used.

Further information: "Notes on the NC program", Page 279

Input

11 MONITORING SECTION START AS "finish contour"	; Start of the monitoring section, specifying an additional name

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING SECTION	Syntax initiator for the monitoring section of process monitoring
START or **STOP**	Start or end of the monitoring section
AS	Additional designation Optional syntax element Only when **START** is selected

Notes

- The control shows the start and end of the monitoring section in the structure.
- End the monitoring section before the end of the program with **MONITORING SECTION STOP**.

  If you do not define an end for the monitoring section, the control ends the monitoring section with **END PGM**.
- Monitoring sections used for process monitoring must not overlap with the **AFC** sections.

  Further information: "Adaptive Feed Control (AFC, option 45)", Page 252

14

Opening CAD Files with the CAD-Viewer

14.1 Fundamentals

Application

CAD-Viewer allows you to open the following standardized file types directly on 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
STL	*.stl	■ Binary ■ ASCII

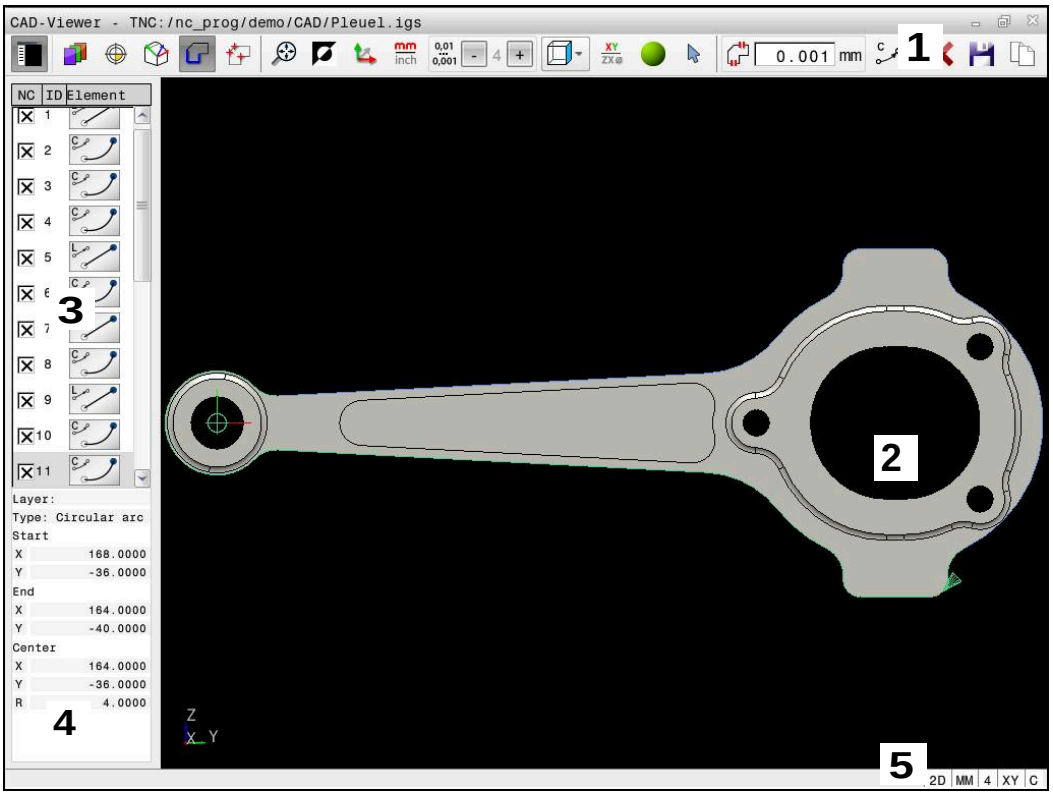
The **CAD-Viewer** runs as a separate application on the third desktop of the control.

Related topics

- Creating 2D sketches on the control
  Further information: User's Manual for Programming and Testing

Description of function

Screen layout



CAD file open in **CAD-Viewer**

CAD Viewer consists of the following areas:

1 Menu bar

Further information: "Menu bar icons", Page 304

2 Graphics window

The CAD model is displayed in the graphics window.

3 List View window

The List View window displays information on the active function (e.g., available layers or the position of the workpiece preset).

4 Element Information window

Further information: "Element Information window", Page 306

5 Status bar

The status bar contains the active settings.

Menu bar icons

The menu bar contains the following icons:

Icon	Function
	**Show sidebar** Show, enlarge, or hide the List view window
	**Display the layer** Display the layer(s) in the List View window **Further information:** "Layer", Page 306
	**Preset** Define the workpiece preset
	Workpiece preset has been defined
	Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD model", Page 307
	**Datum** Set the datum
	Datum has been set **Further information:** "Workpiece datum in the CAD model", Page 310
	**Contour** Select contour (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312
	**Positions** Select drilling positions (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312
	**3D mesh** Create a 3D mesh (option 152) **Further information:** "Generating STL files with 3D mesh (option 152)", Page 318
	**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 mm or inches as the unit of measure Internally, **CAD-Viewer** always uses mm for its calculations. If you select inches as the unit of measure, **CAD-Viewer** will convert all values to inches. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312

Icon	Function
0,01 ... 0,001	**Number of decimal places** Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312 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 (option 50). If you take over a contour or position, the control outputs the NC program in the selected working plane. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312
	Toggle a 3D model between a solid model and a wire-frame model.
	"Select, add, or remove contour elements" mode The icon shows the current mode. Clicking the icon activates the next mode. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 312
	Undo
	**Delete entire list**
	**Save entire list content to a file**
	**Copy entire list contents to clipboard** The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.

Element Information window

In the Element Information window, 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 window, you can use the **Show sidebar** icon to enlarge this window.

With the **Display the layer** icon, the control shows all the layers of the file in the List view window. Use the checkbox in front of the name to show and hide the individual layers.

When you open a CAD file in **CAD-Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.
- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

  Further information: User's Manual for Programming and Testing
- When you select a layer in the list view window, you can use the space bar to show and hide the layer.
- **CAD-Viewer** allows you to open CAD models consisting of any number of triangles.

14.2 Workpiece preset in the CAD model

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

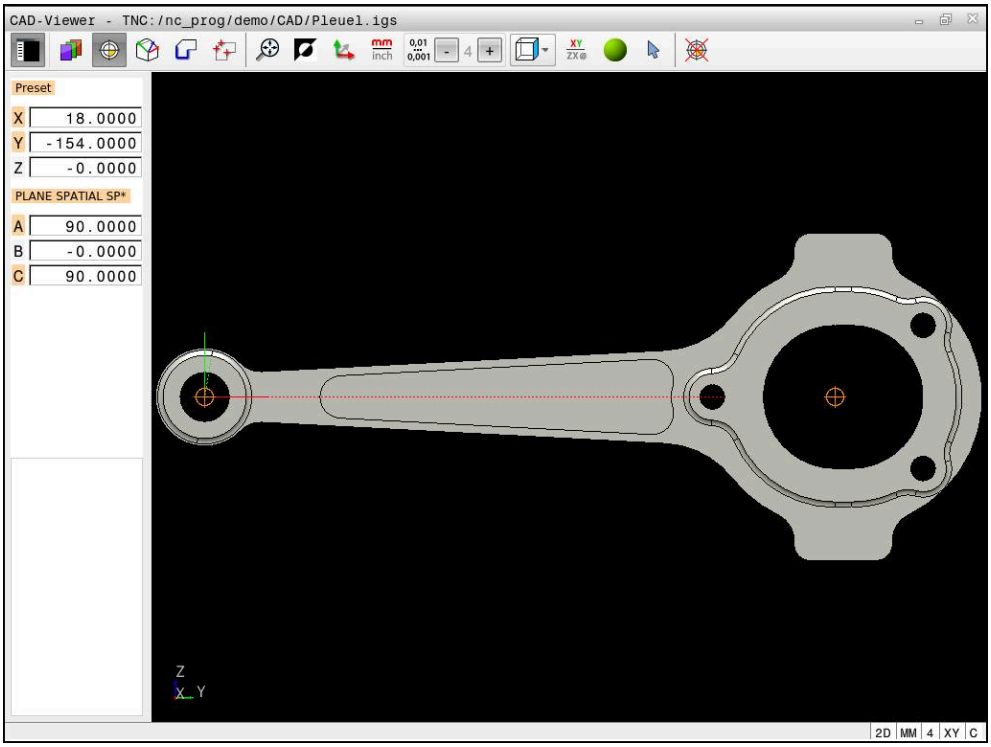
- Presets in the machine
  Further information: "Presets in the machine", Page 154

Description of function

When you select the **Preset** icon, the control displays the following information in the list view window:

- Distance between the defined preset and the drawing datum
- Orientation of the coordinate system with respect to the drawing

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD model

You can position the preset at the following locations:

- By direct input of numerical values into the List View window
- For straight lines:
  - Starting point
  - Midpoint
  - End point
- For circular arcs:
  - Starting point
  - Center point
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center
- At the intersection between:
  - Two straight lines, even if the point of intersection is actually on the extension of one of the lines
  - Straight line and circular arc
  - Straight line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

The control maintains the contents 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 aligning the coordinate system

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 85
- The following contents also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:

- Select **Preset**
- Position the cursor on the desired element
- If you are using a mouse, the control for the element displays selectable presets 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.
- Align the coordinate system, if required

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

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

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

- Select **Preset**
- Click on the first element
- > The control highlights the element in color.
- Click on the second element
- > The control sets the workpiece preset at the intersection of the two elements. The control marks the workpiece preset with a green icon.
- Align the coordinate system, 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.

Aligning the coordinate system

The following conditions must be met in order to orient the coordinate system:

- Preset has been defined
- There are elements next to the preset that can be used for the desired orientation

To align the coordinate system:

- Select an element in the positive direction of the X axis
- > The control aligns the X axis.
- > The control changes the angle **C** in the list view window.
- Select an element in the positive direction of the Y axis
- > The control aligns the Y and Z axes.
- > The control changes the angles **A** and **C** in the list view window.

14.3 Workpiece datum in the CAD model

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 tilting operation.

Related topics

- Presets in the machine
  Further information: "Presets in the machine", Page 154

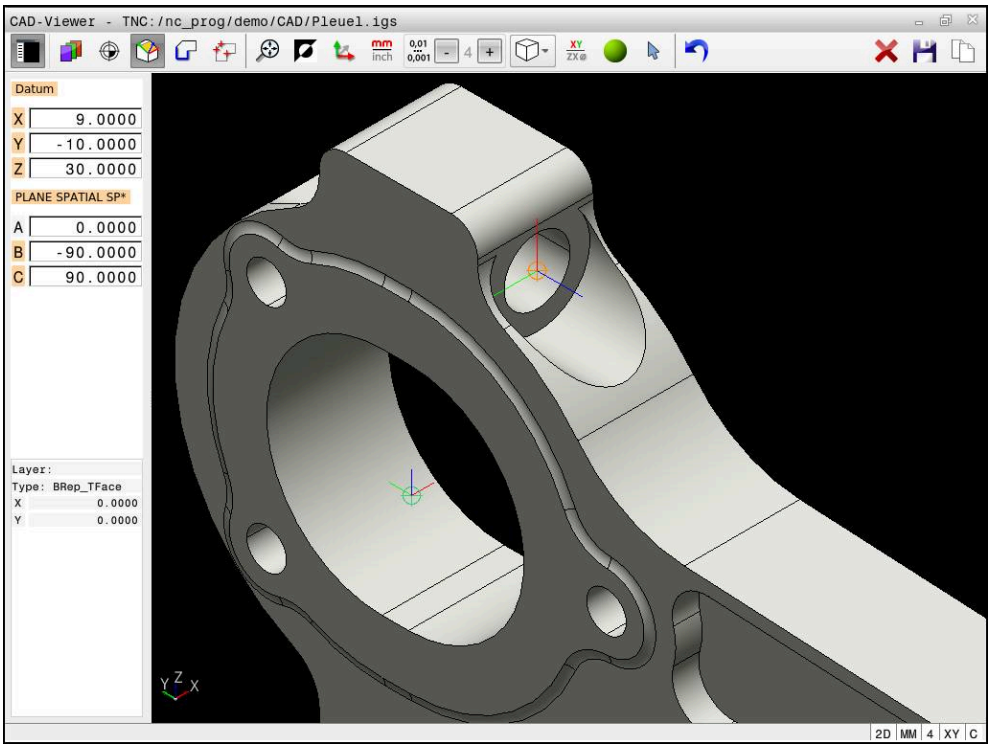
Description of function

When you select the **Datum** icon, the control displays the following information in the list view window:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the coordinate system

You can set a workpiece datum and also move it further by entering values directly in the list view window.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the coordinate system can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD model", Page 307

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 aligning the coordinate system", Page 309

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

You can save the information on the workpiece preset and workpiece datum to a file or to the clipboard without having to resort to CAD Import (software option 42).

The control maintains the contents of the clipboard only as long as **CAD-Viewer** is open.

14.4 Applying contours and positions to NC programs with CAD Import (option 42)

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

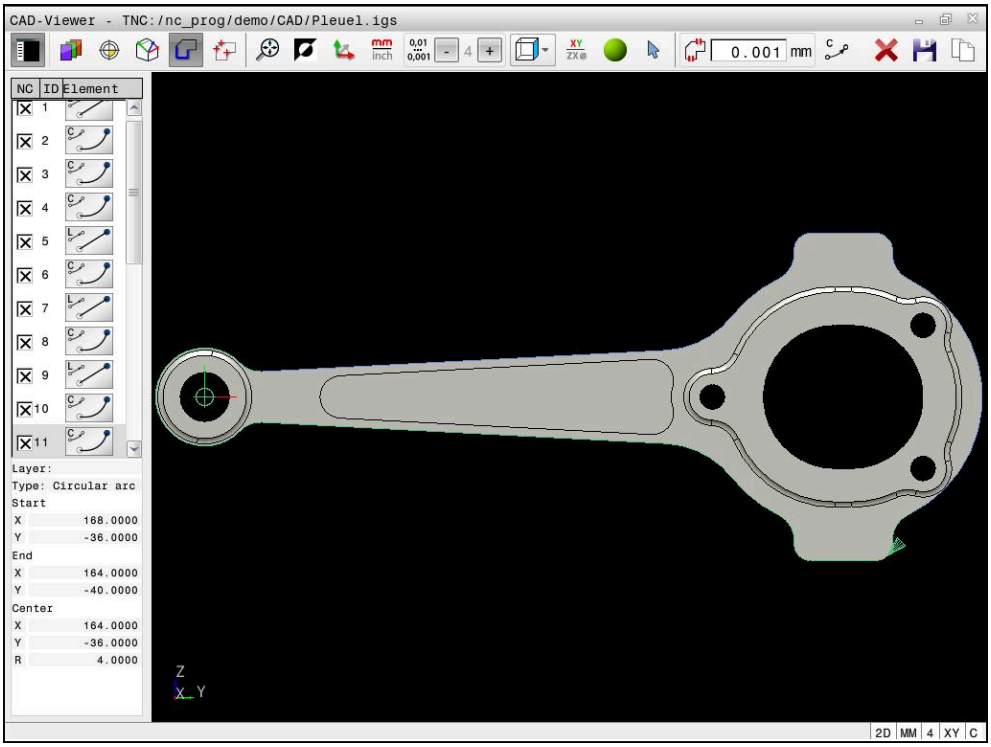
Requirement

- CAD Import (software option 42)

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 584



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	Function
	**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 when the drawing was made. The default setting is 0.001 mm
 	**C** or **CR** Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.
	**Show connections between two positions** Specify whether the control should display the tool path as a dashed line during selection of machining positions
	**Apply path optimization** The control optimizes the tool traverse movement so that there are shorter traverse distances between the machining positions. You reset this optimization by selecting the icon again
	**Find circles according to diameter range. Load center coordinates to the position list** The control opens a pop-up window in which you can filter holes (full circles) based on their size

Applying contours

The following elements can be selected as a contour:

- Line segment
- Circle
- Circular arc
- Polyline
- Any curves (e.g., splines, ellipses)

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.

Linearization

During linearization, a contour is divided into individual positions. The CAD Import creates a straight line **L** for each position. With the CAD Import, you can therefore also apply contours that cannot be programmed with the path functions of the control (e.g., splines).

The **CAD-Viewer** linearizes all of the contours that are not in the XY plane. The higher the resolution, the more accurately the control displays the contours.

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 316

The following file types are available:

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

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

CAD-Viewer also recognizes circles that consist of two semi-circles as machining positions.

Multi-selection filter settings

After you have used the quick selection function to mark drilling positions, a pop-up window appears, showing the smallest diameter found to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can transfer the hole diameters that you want.

The following buttons are available:

Icon	Filter setting for the smallest diameter
I<<	Display the smallest diameter found (default setting)
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter

Icon	Filter setting of largest diameter
<<	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>I	Display the largest diameter found (default setting)

14.4.1 Selecting and saving a contour

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 85
- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.

Selecting a contour with existing contour elements

To select and save a contour with existing contour elements:

- Select **Contour**
- Place the cursor on the first contour element
- The control shows the suggested direction of rotation as a dashed line.
- If necessary, move the cursor towards the more distant end point.
- The control changes the suggested direction of rotation.
- Select the contour element
- The selected contour element is displayed in blue and is marked in the Sidebar window.
- 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 Sidebar window.

- Select **Save entire list content to a file**
- The **Define file name for contour program** window opens.
- Enter the desired name
- Select the path to the storage location
- Select **Save**
- The selected contour is saved as an NC program.

- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you select an element with the CTRL key pressed, it is deselected for export.

Selecting paths independent of existing contour elements

To select a path independent of existing contour elements:

- Select **Contour**

- Select **Select**
- > 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 the contour as a workpiece blank definition (option 50)

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 these steps with touch gestures.
  Further information: "Common gestures for the touchscreen", Page 85
- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.
  Further information: "Selecting and saving a contour", Page 315

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 window.

Multiple selection by area

To select multiple positions within an area:

- Select **Positions**

- Select **Select**
- > The icon changes, and the control activates the **Add** mode.
- Drag a box around the area while holding down the left mouse button
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameters found.
- Change the filter settings as needed
- Press **OK**
- > The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- > The control shows the traversing distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:

- Select **Positions**

- **Find circles according to diameter range.** Select **Find circles according to diameter range. Load center coordinates to the position list**
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.

Notes

- Set the correct unit of measure to make sure that **CAD-Viewer** displays 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 maintains the contents of the clipboard only as long as **CAD-Viewer** is open.
- **CAD-Viewer** also recognizes circles that consist of two semi-circles as machining positions.
- 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.

Notes on applying contours

- If you double-click a layer in the list view window, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved it on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
  As an alternative, you can switch to the Remove mode:

14.5 Generating STL files with 3D mesh (option 152)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective files of fixtures and tool holders, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture Monitoring (option 40)
- Export simulated workpiece as STL file
- Using an STL file as workpiece blank
  Further information: User's Manual for Programming and Testing

Requirement

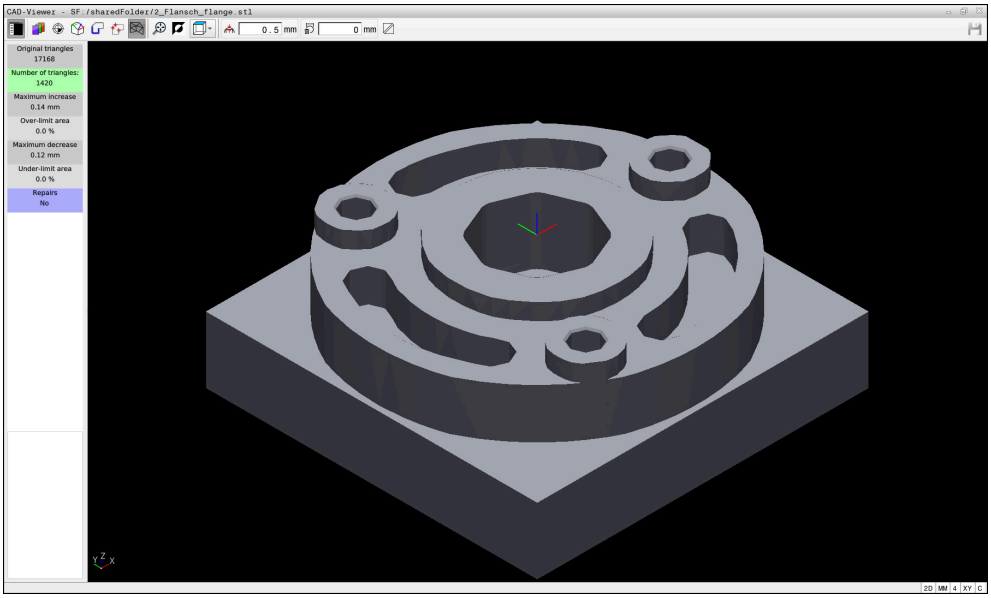
- CAD Model Optimizer (software option 152)

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 selected settings in **3D mesh** mode.

The Sidebar window 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 320
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

Option	Meaning
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control indicates the type of repair (e.g., **Hole Int Shells**). This indication consists of the following items: ■ **Hole** **CAD-Viewer** closed holes in the 3D model. ■ **Int** **CAD-Viewer** removed self-intersections. ■ **Shells** **CAD-Viewer** joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:

- Max. 20 000 triangles
- Triangular mesh forms a closed shell

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD-Viewer provides the following functions:

Symbol	Function
	**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

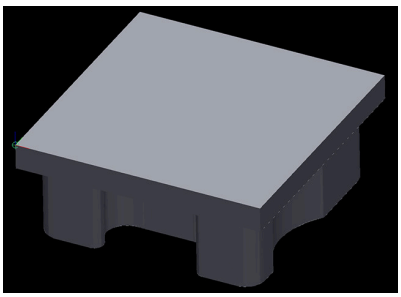
To position an STL file for rear-face machining:

- Export the simulated workpiece as an STL file

  Further information: User's Manual for Programming and Testing

- Select the **Files** operating mode
- Select the exported STL file
- > The control opens the CAD files in **CAD-Viewer**.

- Select **Preset**
- > In the Sidebar window, the control displays information on the position of the preset.
- Enter the value of the new preset under **Preset** (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 320

- Select **Save**
- > The control opens the **Define file name for 3D mesh** menu.
- 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: User's Manual for Programming and Testing

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

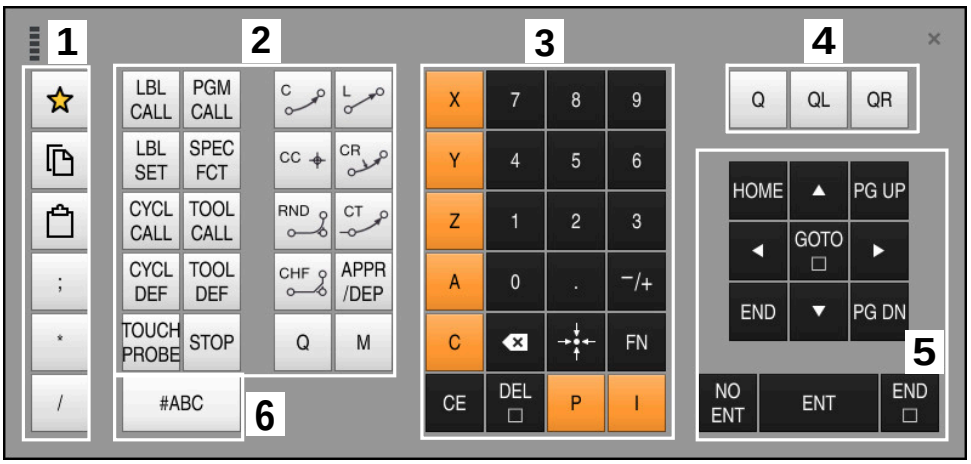
The control opens NC input mode by default after the start procedure.

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.

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**

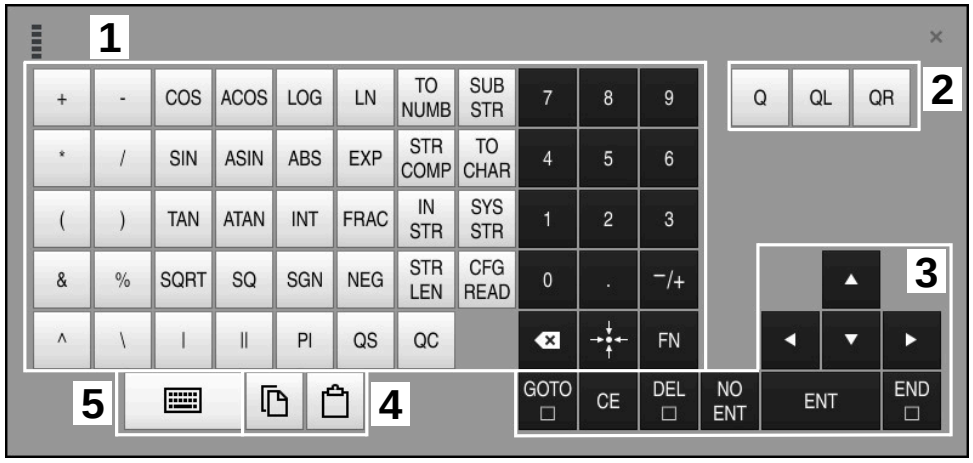
Areas of text input

Virtual keyboard in text input mode

The text input contains the following areas:

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

Areas of formula input



Virtual keyboard in formula input mode

The formula input contains the following areas:

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

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.
	Reset error	The control must be restarted. This message cannot be cleared.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content

Collapsed message menu on the information bar

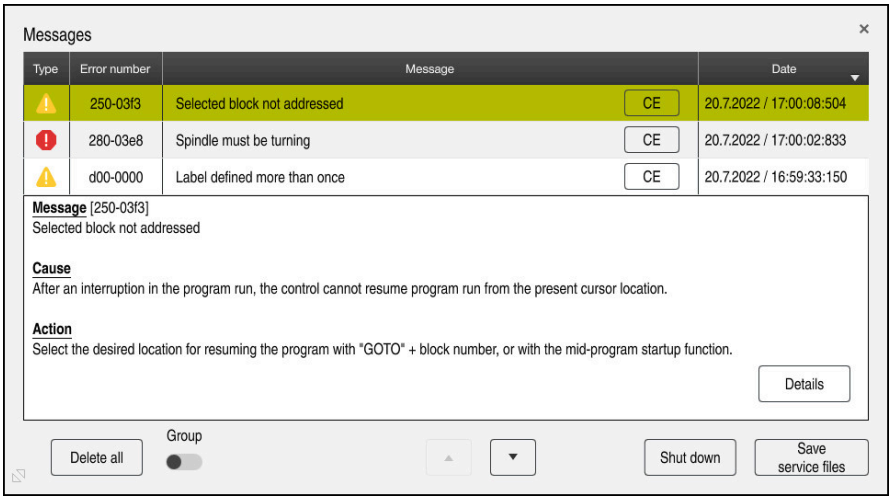
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (root cause, correction, information on the NC program)

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** toggle switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

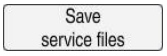
- If an error occurs, you can create a service file manually.
  Further information: "Creating a service file manually", Page 330
- If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.
  Further information: "Creating a service file automatically", Page 330

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.

15.2.1 Creating a service file manually

To create a service file manually:

- ▶ Expand the message menu



- ▶ Select **Save service files**
- > The control opens the **Save service file** window.
- ▶ Enter the file name

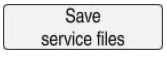
- ▶ Press **OK**
- > The control saves the service file in the **TNC:\service** directory.

15.2.2 Creating a service file automatically

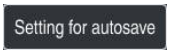
You can specify up to five error numbers for which the control will automatically create a service file if one of these errors occurs.

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
- ▶ Activate the **Active** check box
- > If the error occurs, the control automatically creates a service file.
- ▶ Enter a comment, if applicable (e.g., to describe the problem)

16

Touch Probe Functions in the Manual Operating Mode

16.1 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
  Further information: User's Manual for Measuring Cycles for Workpieces and Tools
- Preset table
  Further information: "Preset table", Page 443
- Datum table
  Further information: User's Manual for Programming and Testing
- Reference systems
  Further information: "Reference systems", Page 198
- Preassigned variables
  Further information: User's Manual for Programming and Testing

Requirements

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 346

Description of function

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

- Define the workpiece preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- Measure the tool

Within the functions, the control provides the following probing methods:

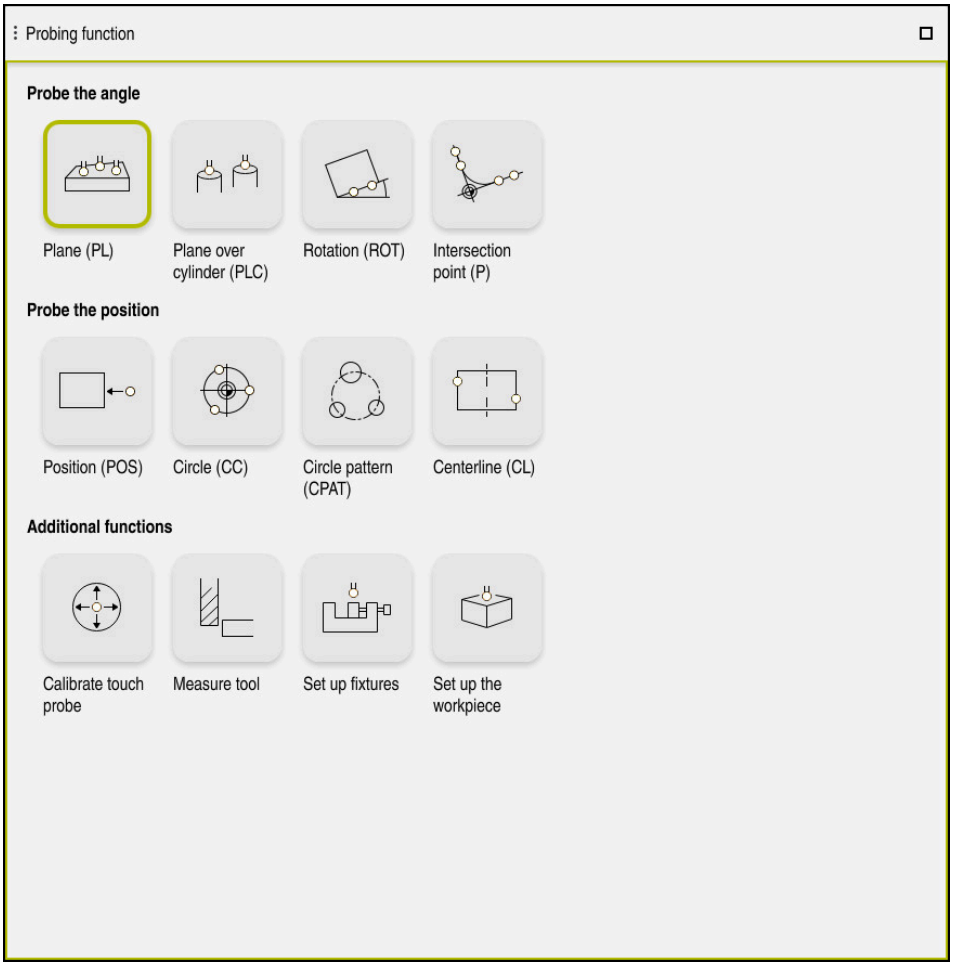
- Manual probing method

  You position and start individual probing processes manually within a touch probe function.

  Further information: "Setting a preset in a linear axis", Page 339
- Automatic probing method

  You manually position the touch probe to the first probing point before the start of the probing routine and fill out a form with the individual parameters for the respective touch probe function. When you start the touch probe function, the control automatically positions and automatically performs probing.

  Further information: "Determining the circle center point of a stud using the automatic probing method ", Page 341



Probing function workspace

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the **Plane (PL)** function to determine the solid angle of a plane. You then save the values in the preset table or align the plane.
Plane over cylinder (PLC) 	Use the **Plane over cylinder (PLC)** function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table or align the plane.
Rotation (ROT) 	Use the **Rotation (ROT)** function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. **Further information:** "Determining and compensating the rotation of a workpiece", Page 343
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.

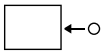
Further information: "Preset table", Page 443

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 354

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 339
Circle (CC) 	The **Circle (CC)** function is used to determine the coordinates of a circle center point (e.g., for a hole or for a stud). **Further information:** "Determining the circle center point of a stud using the automatic probing method ", Page 341
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 group

The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe 	The **Calibrate touch probe** function is used to determine the length and radius of a workpiece touch probe. **Further information:** "Calibrating the workpiece touch probe", Page 346
Measure tool 	The **Measure tool** function allows you to measure tools by scratching. In this function, the control supports milling tools, drilling tools and turning tools.
Set up fixtures 	The **Set up fixtures** function is used to determine the position of a fixture in the machine's working space using a workpiece touch probe (option 140). **Further information:** "Integrating the fixtures into collision monitoring (option 140)", Page 235
Set up the workpiece 	The **Set up the workpiece** function is used to determine the position of a workpiece in the machine's working space using a workpiece touch probe (option 159). **Further information:** "Setting up the workpiece with graphical support (option 159)", Page 356

Buttons

General buttons in the touch probe functions

The following buttons are available, depending on the selected touch probe function:

Button	Function
	Finish the active touch probe function
	Select the workpiece preset and the pallet presets and edit values if required **Further information:** "Change the preset window", Page 338 **Further information:** "Preset table", Page 443 The control grays out this icon while a probing process takes place. In this condition, you can check the presets but you cannot edit them. You need to stop the probing process in order to edit presets.
	Display help graphics for the selected touch probe function
	Select the probing direction
	Apply the actual position
	Approaching and probing points on a straight surface manually
	Approaching and probing points on a stud or in a hole manually
	Approaching and probing points on a stud or in a hole automatically After the last touching process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.

Calibration buttons

The control offers the following functions for calibrating a 3D touch probe:

Button	Function
	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 346

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Button	Function
	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 Optionally calibrating the tool touch probe in 3D (option 92) **Further information:** User's Manual for Programming and Testing **Further information:** "3D calibration (option 92)", Page 347

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **3-D rotation** window, the control opens the **Working plane is inconsistent!** window.

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Function
3-D ROT Apply status	The **3-D ROT Apply status** function transfers the position of the rotary axes into the **3-D rotation** window. **Further information:** "3-D rotation window (option 8)", Page 220
3-D ROT Ignore status	The **3-D ROT Ignore status** function makes the control calculate the probing results, assuming that the rotary axes are in their zero position.
Align the rotary axes	The **Align the rotary axes** function aligns the rotary axes to the active tilting situation in the **3-D rotation** window.

Buttons for measured values

After executing a touch probe function, you select the desired control reaction.

The control offers the following functions:

Button	Function
Compensate the active preset	The **Compensate the active preset** function transfers the measuring result into the active line of the preset table. **Further information:** "Preset table", Page 443
Write the datum	The **Write the datum** function transfers the measuring result into a desired line of the datum table. **Further information:** User's Manual for Programming and Testing
Align rotary table	The **Align rotary table** function aligns the rotary axes mechanically according to the measuring result.

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 213

The **Change the preset** window provides the following buttons:

Button	Meaning
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	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.

Log file of touch probe cycles

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

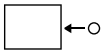
If you run several touch probes cycles in a row, the control stores the measured values below each other.

16.1.1 Setting a preset in a linear axis

To probe the preset in any axis:

- Select the **Manual** operating mode

- Call the workpiece touch probe as a tool
- Select the **Setup** application



- Select the **Position (POS)** touch probe function
- > The control opens the **Position (POS)** touch probe function.

- Select **Change the preset**
- > The control opens the **Change the preset** window.
- Select the desired row of the preset table
- > The control highlights the selected line in green.

- Press **Apply**
- > The control activates the selected line as the workpiece preset.
- Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)

- Select the probing direction (e.g., **Z-**)

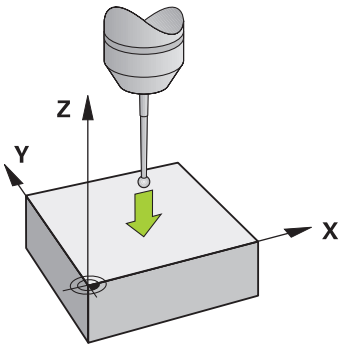
- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)

Compensate the active preset

- Select **Compensate the active preset**
> The control enters the defined nominal value in the preset table.
> The control marks the row with an icon.

> If you use the **Write the datum** function, the control also marks this row with an icon.
>
> When you have completed the probing process in the first axis, you can probe up to two additional axes using the **Position (POS)** probing function.

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



16.1.2 Determining the circle center point of a stud using the automatic probing method

To probe a circle center point:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool
  Further information: "Manual operation application", Page 146
- Select the **Setup** application

- Select **Circle (CC)**
- > The control opens the **Circle (CC)** probing function.

- If necessary, select another preset for the probing process

- Select measuring method **A**

- Select **Type of contour** (e.g., stud)
- Enter **Diameter** (e.g., 60 mm)
- Enter **Starting angle** (e.g., –180°)
- Enter **Angular length** (e.g., 360°)
- Position the 3D touch probe at the desired probing position next to the workpiece and below the workpiece surface

- Select the probing direction (e.g., **X+**)
- Turn the feed rate potentiometer to zero

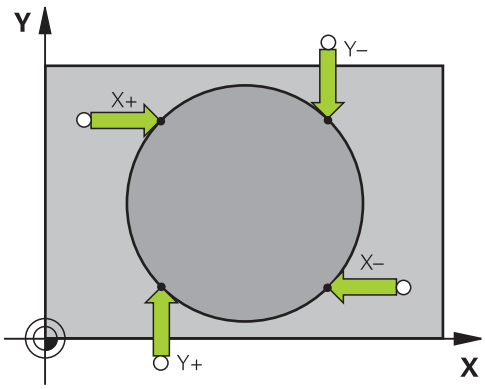
- Press the **NC start** key
- Slowly turn on the feed rate potentiometer
- > The control executes the touch probe function based on the data entered.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the scanned axes (e.g., **0**)

Compensate the active preset

- ▶ Select **Compensate the active preset**
- > The control sets the preset to the entered nominal value.
- > The control marks the row with an icon.

> If you use the **Write the datum** function, the control also marks this row with an icon.

- ▶ Select **Exit probing**
- > The control closes the **Circle (CC)** probing function.



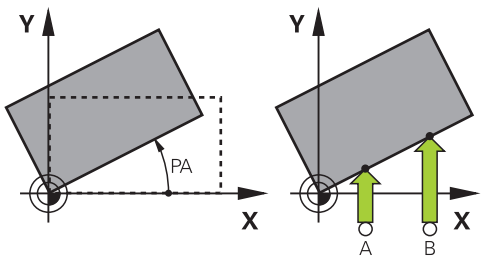
16.1.3 Determining and compensating the rotation of a workpiece

To probe the rotation of a workpiece:

- Select the **Manual** operating mode
- Call the 3D touch probe as a tool
- Select the **Setup** application
- Select **Rotation (ROT)**
- > The control opens the **Rotation (ROT)** probing function.
- If necessary, select another preset for the probing process
- Position the 3D touch probe at the desired probing position in the workspace
- Select the probing direction (e.g., **Y+**)
- Press the **NC start** key
- > The control executes the first probing process and limits the subsequently selectable probing directions.
- Position the 3D touch probe at the second probing position in the workspace
- Press the **NC start** key
- > The control executes the probing process and then shows the measurement results.
- Select **Compensate the active preset**
- > The control transfers the determined basic rotation to the **SPC** column of the active line of the preset table.
- > The control marks the row with an icon.

> Depending on the tool axis, the measurement result can also be written to another column of the preset table (e.g., **SPA**).

- Select **Exit probing**
- > The control closes the **Rotation (ROT)** probing function.



16.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can use all manual touch probe functions with manual probing methods with mechanical buttons or with scratching.

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

To determine a basic rotation with a mechanical probe:

- Select the **Manual** operating mode

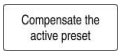
- Insert the tool, such as an analog 3D probe or feeler lever gage
- Select the **Setup** application

- Select the **Rotation (ROT)** probing function

- Select the probing direction (e.g., **Y+**)
- Move the mechanical probe to the first position to be captured by the control.

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

- Select **Accept position**
> The control saves the current position.



- Select **Compensate the active preset**
> The control transfers the determined basic rotation to the active line of the preset table.

> The control marks the row with an icon.

> The determined angles have different effects depending on whether they are transferred as an offset or as a basic rotation to the corresponding table.
>
> **Further information:** "Comparison of offset and 3D basic rotation", Page 354

- Select **Exit probing**
> The control closes the **Rotation (ROT)** probing function.

Notes

- When you use a non-contacting tool touch probe, use touch probe, then you are using touch probe functions from the third-party manufacturer. This is the case, for example, with a laser touch probe. Refer to your machine manual.
- The accessibility of the pallet preset table in the touch probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

  Further information: "Global Program Settings (GPS, option 44)", Page 261
- You can use the manual touch probe functions only with restrictions in turning mode (option 50).
- 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 164
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no tool touch probe is inserted, the actual position can be captured with **NC START**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
  - Initial configuration
  - Broken stylus
  - Stylus replacement
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis

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.

16.2 Calibrating the workpiece touch probe

Application

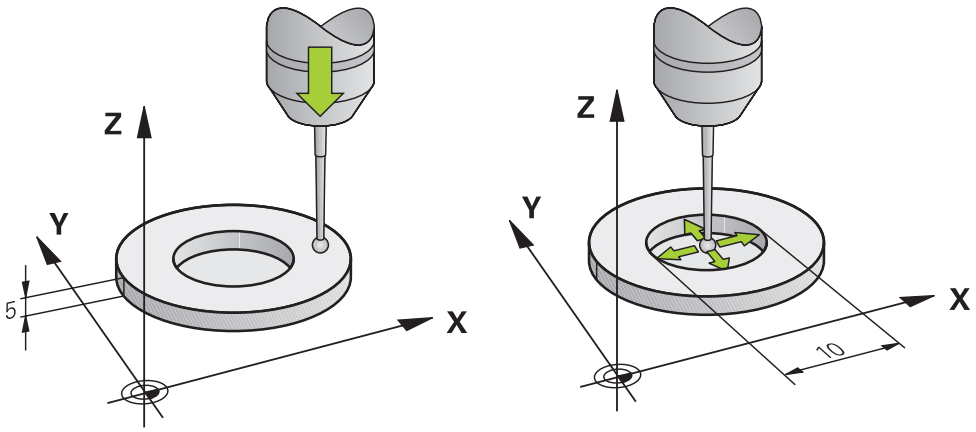
In order to precisely specify the actual trigger point of a 3D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.

During 3D calibration, you determine the angle-dependent deflection behavior of a workpiece touch probe in any probing direction (option 92).

Related topics

- Calibrate the workpiece touch probe automatically
  Further information: User's Manual for Measuring Cycles for Workpieces and Tools
- Touch probe table
  Further information: "Touch probe table tchprobe.tp", Page 431
- 3D radius compensation depending on the contact angle (option 92)
  Further information: User's Manual for Programming and Testing

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 159

You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

At first, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis For this, manually position the extension in the direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp", Page 431

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

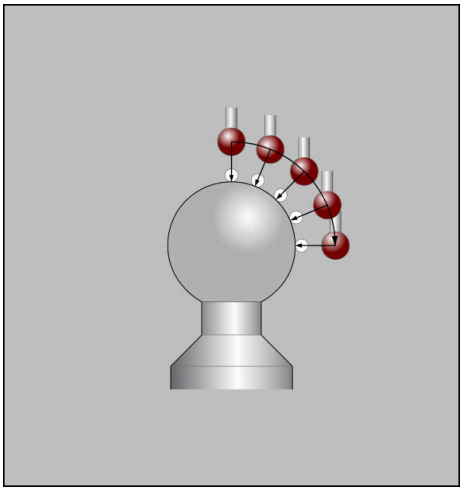
While calibrating the touch probe, ensure that the feed rate override is 100%. That way you can always use the same feed rate for the subsequent probing processes as was used for the calibration. Hence, you can exclude inaccuracies during the probing caused by modified feed rates.

3D calibration (option 92)

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.

16.2.1 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 214
- 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., with **-0.5 mm**)
- Overmill the workpiece blank using the handwheel
- Set the preset again in the tool axis (e.g., with **Z=0**)
- Switch off spindle (e.g., with **M5**)
- Replace the tool touch probe
- Select the **Setup** application

- Select **Calibrate touch probe**

- Select the **Length calibration** measurement method

> The control displays the current calibration data.

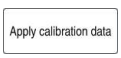
- Enter the reference surface position (e.g., with **0**)
- Position the workpiece touch probe close to the surface of the overmilled area

> Check that the area to be probed is flat and free of chips before you start the touch probe function.

- Press the **NC start** key

> The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.

- Check results



- Select **Apply calibration data**

> The control transfers the calibrated length of the 3D touch probe to the tool table.

- Select **Exit probing**

> The control closes the **Calibrate touch probe** function.

16.2.2 Calibrating the radius of the workpiece touch probe

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

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

- Select the **Manual** operating mode
- Position the 3D touch probe in the hole of the setting ring

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

- Select the **Setup** application

- Select **Calibrate touch probe**

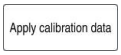
- Select **Radius** measurement method

- Select **Setting ring** calibration standard
- Enter the diameter of the ring gauge
- Enter the start angle
- Enter the number of touch points

- Press the **NC Start** key

> The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

- Check results



- Select **Apply calibration data**

> The control stores the calibrated radius of the 3D touch probe in the tool table.

- Select **Exit probing**

> The control closes the **Calibrate touch probe** function.

16.2.3 3D calibration of workpiece touch probe (option 92)

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

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

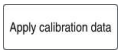
- Select the **Manual** operating mode
- Position the workpiece touch probe centrally above the sphere
- Select the **Setup** application

- Select **Calibrate touch probe**

- Select **Radius** measurement method

- Select the **Calibration sphere** calibration standard
- Enter the diameter of the sphere
- Enter the start angle
- Enter the number of touch points

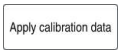
- Press the **NC Start** key
- The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results



- Select **Apply calibration data**
- The control stores the calibrated radius of the 3D touch probe in the tool table.
- The control shows the **3D calibration** measurement method.

- Select the **3D calibration** measurement method
- Enter the number of touch points

- Press the **NC Start** key
- The 3D touch probe probes all required touch points in an automatic probing routine.



- Select **Apply calibration data**
- The control saves the deviations in a compensation value table under **TNC:\system\3D-ToolComp**.

- Select **Exit probing**
- The control closes the **Calibrate touch probe** function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- 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 431

16.3 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.

16.3.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:

- Select the **Manual** operating mode
- Select **Suppress touch probe monitoring**
- The control disables touch-probe monitoring for 30 seconds.
- If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

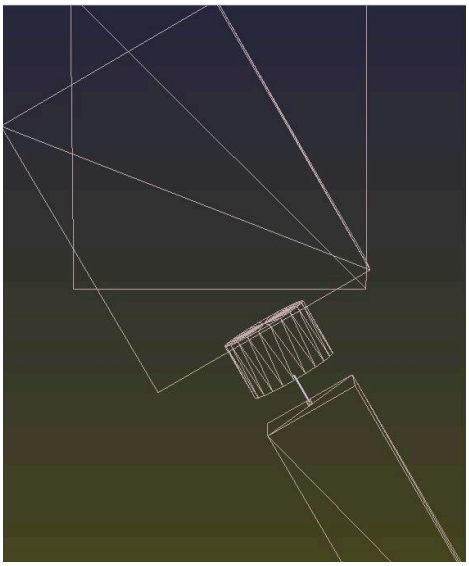
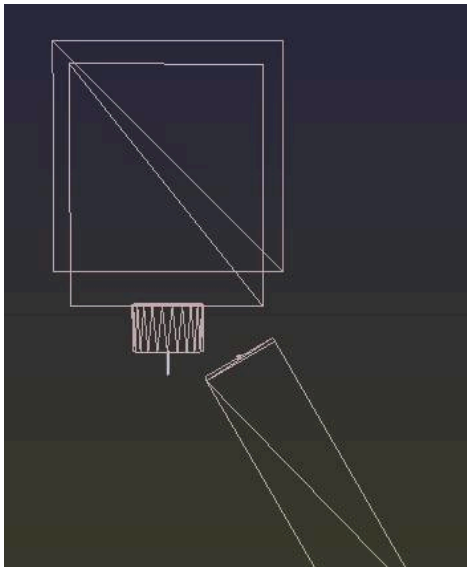
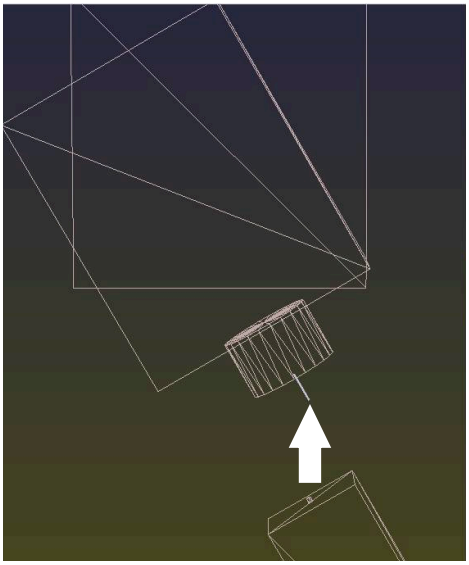
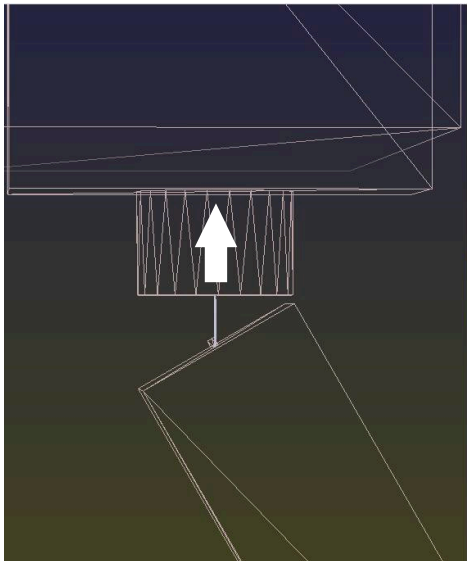
While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

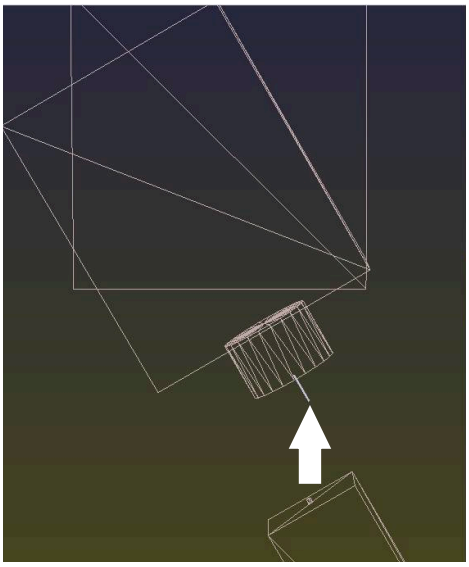
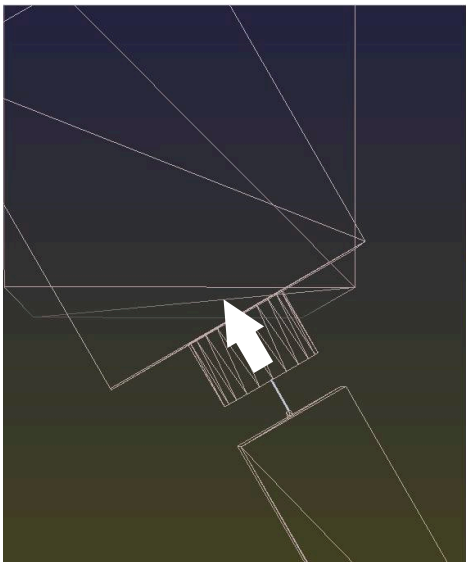
- Carefully move the axes in the **Manual** operating mode

If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

16.4 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.

16.5 Setting up the workpiece with graphical support (option 159)

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. Tilting and probing on curved surfaces can be performed during setup in order to probe also complex workpieces, such as free-form parts.

The control supports you additionally by showing the clamping 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 331
- Generating an STL file of a workpiece

  Further information: User's Manual for Programming and Testing
- **Simulation** workspace

  Further information: User's Manual for Programming and Testing
- Calibrating fixtures with graphical support (option 140)

  Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 235

Requirements

- Advanced Functions Set 2 (software option 9)
- Model Aided Setup (software option 159)
- Touch probe properly defined in the tool management:
  - Spherical radius in the **R2** column
  - If probing on inclined surfaces, the spindle tracking in the **TRACK** column needs to be active

  Further information: "Tool data for touch probes", Page 182
- Workpiece touch probe calibrated

  If probing on inclined surfaces, a 3D calibration of the touch probe needs to be performed (option 92).

  Further information: "Calibrating the workpiece touch probe", Page 346
- 3D model of the workpiece as STL file

  The STL file may contain up to 300,000 triangles. The more the 3D model corresponds to the actual workpiece, the higher the possible workpiece setup accuracy.

  If applicable, optimize the 3D model with the **3D mesh** function (option 152).

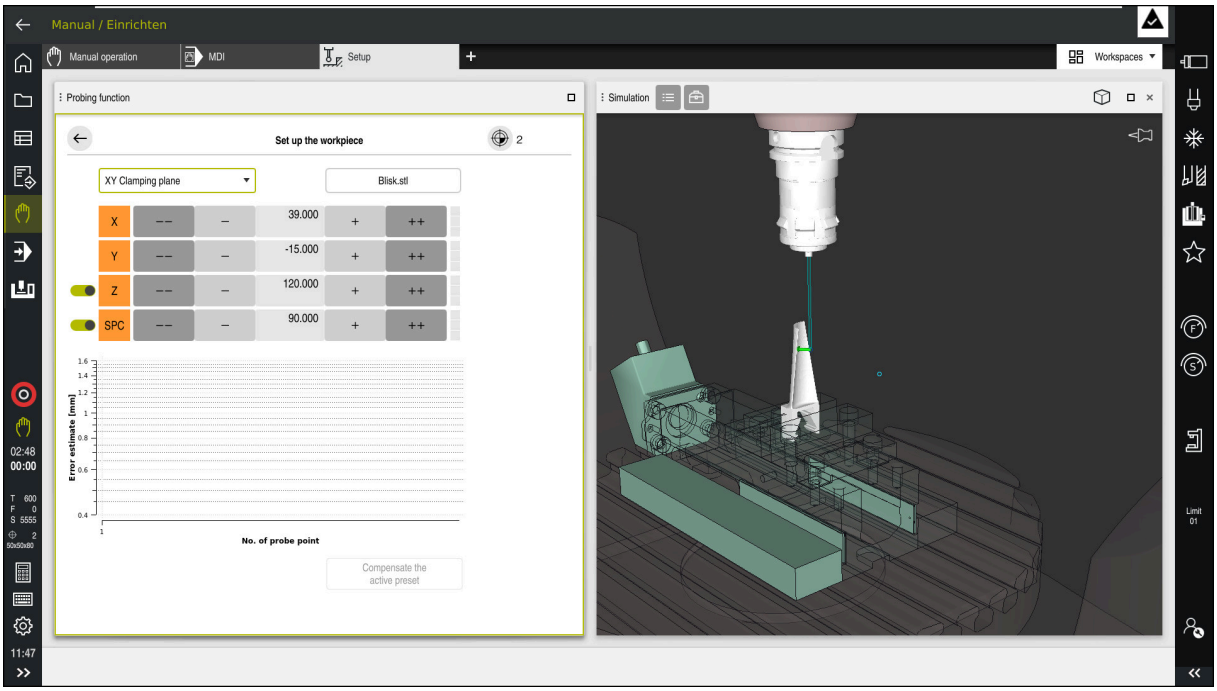
  Further information: User's Manual for Programming and Testing

Description of function

The **Set up the workpiece** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

Extensions of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphical support for setting up the workpiece.



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.

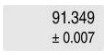
    > Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    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	Function
	Open the **Change the preset** window You can select the workpiece preset and the pallet preset and edit values if required. After the first point has been probed, the control grays out 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 359
	File name of 3D model
	Shifts the position of the virtual workpiece by 10 mm or 10° in the negative axis direction Shifts the workpiece in mm in a linear axis and in degrees in a rotary axis.
	Shifts the position of the virtual workpiece by 1 mm or 1° in the negative axis direction
	■ Enter the position of the virtual workpiece directly ■ Value and estimated accuracy of the value after the probing
	Shifts the position of the virtual workpiece by 1 mm or 1° in the positive axis direction
	Shifts the position of the virtual workpiece by 10 mm or 10° in the positive axis direction
    	Status of direction The control displays the following colors: ■ Gray The axis direction is deselected in this setup process and is not considered. ■ White No touch points have been determined yet. ■ Red The control cannot locate the workpiece position in this axis direction. ■ Yellow The position of the workpiece in this axis already contains information. The information is not meaningful yet. ■ Green The control can locate the workpiece position in this axis direction.
Compensate the active preset	The control saves the determined values in the active row of the preset table.

Probing mode

The following modes for probing the workpiece are available to you:

- **XY Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPC**
- **XZ Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPB**
- **YZ Clamping plane**
  X, **Y** and **Z** axis directions as well as spatial angle **SPA**
- **6D**
  X, **Y** and **Z** axis directions as well as spatial angles **SPA**, **SPB** and **SPC**

Depending on the probing mode, the control displays the respective axis directions and spatial angles. In the **XY**, **XZ** and **YZ** clamping planes a toggle switch allows you to deselect the respective tool axis and spatial angle, if required. The control will not take deselected axis directions into account in the setup process and positions the workpiece by considering the remaining axis directions only.

HEIDENHAIN recommends executing the setup process as follows:

1 Pre-position a 3D model in the machine's working space

  At this point in time, the control does not know the precise position of the workpiece, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe produces values close to the position of the real workpiece.

2 Set the first touch points in the **X**, **Y** and **Z** axis directions

  If the control can determine the position in one axis direction, it will change the status of that axis to green.

3 Determine the spatial angle by setting further touch points

  To achieve maximum accuracy when probing the spatial angles, the touch points should be as far apart from one another as possible.

4 Increase the accuracies by additional check points

  Additional check points at the end of the measuring process improve the matching accuracy and minimize the misalignment between the 3D model and the real workpiece. Perform as many probing processes as necessary until the control displays the desired accuracy beneath the current value.

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

Further information: "Error estimate diagram", Page 360

Error estimate diagram

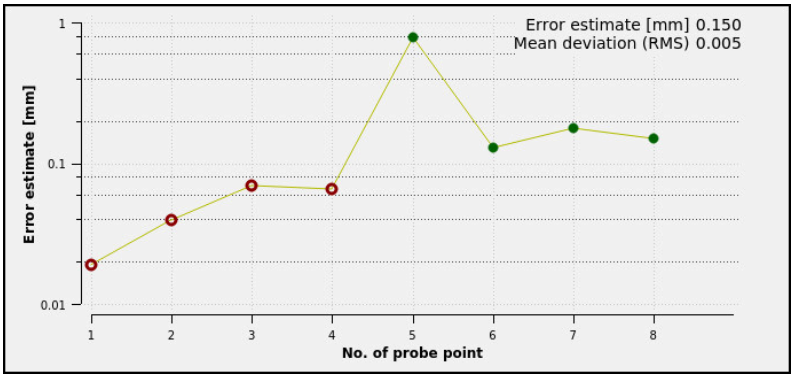
Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine.

The error estimate diagram shows the estimated value of the distance of the 3D model from the real workpiece. For this purpose, the control considers not only the touch points, but the whole workpiece.

When the error estimate diagram shows green circles and the desired accuracy, the setup process will be complete.

The following factors influence the accuracy that can be achieved when measuring workpieces:

- Accuracy of workpiece touch probe
- Accuracy of the machine kinematic configuration
- Deviations of the 3D model from the real workpiece
- Condition of the actual workpiece (e.g., unmachined areas)



Error estimate diagram in the **Set up the workpiece** function

The error estimate diagram of the **Set up the workpiece** function shows the following information:

- **Mean deviation (RMS)**

  This area shows the average distance of the real workpiece from the 3D model in mm.
- **Error estimate [mm]**

  This axis shows the course of the error estimate based on the individual touch points. The control shows red circles until it can determine all axis directions. From then on the control will show green circles.
- **No. of probe point**

  This axis shows the numbers of the individual probing points.

16.5.1 Setting up the workpiece

Use the **Set up the workpiece** function to set the preset:

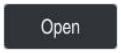
- Affix a real workpiece in the machine's working space

- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the workpiece at a notable point (e.g., a corner)

> This step makes the subsequent steps easier.

- Select the **Setup** application

- Select **Set up the workpiece**
- > The control opens the **Set up the workpiece** menu.
- Select a 3D model matching the real workpiece



- Select **Open**
- > The control opens the selected 3D model in the simulation.

- If necessary, open the **Change the preset** window
- Select a new preset if necessary

- Select the **Apply** function if necessary

- Pre-position the 3D model by using the buttons for the individual axis directions within the virtual working space of the machine

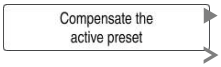
> For pre-positioning the workpiece, use the workpiece touch probe as a point of reference.
>
> Even during the setup process, the shift functions are available for correcting the fixture position manually. Then, probe a new point.

- Define 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**



- > 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.

17

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.

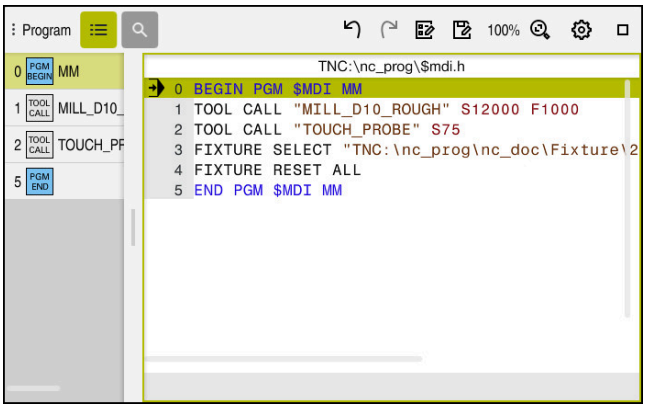
You can also create an NC program step by step. The control remembers modally effective program information.

Related topics

- Creating NC programs
  Further information: User's Manual for Programming and Testing
- Running NC programs
  Further information: "Program Run", Page 367

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**.



Program workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **GS** (option 44)
  Further information: "Global Program Settings (GPS, option 44)", Page 261
- **Help**
- **Positions**
  Further information: "Positions workspace", Page 109
- **Program**
  Further information: User's Manual for Programming and Testing
- **Simulation**
  Further information: User's Manual for Programming and Testing
- **Status**
  Further information: "Status workspace", Page 117
- **Keyboard**
  Further information: "Virtual keyboard of the control bar", Page 324

Buttons

In the **MDI** application, the function bar provides the following buttons:

Button	Meaning
Klartext programming	If this toggle switch is active, then you are using dialog-guided programming. If this toggle switch is not active, then you are programming in the text editor. **Further information:** User's Manual for Programming and Testing
Insert NC function	The control opens the **Insert NC function** window. **Further information:** User's Manual for Programming and Testing
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:** User's Manual for Programming and Testing
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** User's Manual for Programming and Testing
/ Skip block off/on	Hide NC blocks with a **/** 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:** User's Manual for Programming and Testing
Skip block	If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** User's Manual for Programming and Testing
; 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:** User's Manual for Programming and Testing
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 372
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 486
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC, option 145)", Page 260
Editing	The control opens the context menu. **Further information:** User's Manual for Programming and Testing
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 373
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 effective program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always run the NC blocks in **Single Block** mode. After the control has run an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 373

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 window **Q parameter list**
  - 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 an NC program with **PGM CALL**, **SEL PGM**, or **CALL SELECTED PGM**
  - Test Run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** while program run is interrupted
  - **Block scan** function

18

Program Run

18.1 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 one block at a time or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
  Further information: "The MDI Application ", Page 363
- Creating NC programs
  Further information: User's Manual for Programming and Testing
- Pallet tables
  Further information: User's Manual for Programming and Testing

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

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 379

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 the **Single Block** mode, you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the TNC bar", Page 115

The **Program Run** operating mode provides the following workspaces:

- **GS** (option 44)
  Further information: "Global Program Settings (GPS, option 44)", Page 261
- **Positions**
  Further information: "Positions workspace", Page 109
- **Program**
  Further information: User's Manual for Programming and Testing
- **Simulation**
  Further information: User's Manual for Programming and Testing
- **Status**
  Further information: "Status workspace", Page 117
- **Process Monitoring**
  Further information: User's Manual for Programming and Testing

When you open a pallet table, the control will display the **Job list** workspace. You cannot modify this workspace.

Further information: User's Manual for Programming and Testing

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. If 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 run or is marked for running.
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:** User's Manual for Programming and Testing
Compensation tables	The control opens a selection menu with the following tables: ■ **D** ■ **T-CS** ■ **WPL-CS** **Further information:** "Compensation during program run", Page 388
GOTO Cursor	The control marks the currently selected table row for execution. Active only if a pallet table is open (option 22) **Further information:** User's Manual for Programming and Testing
F limited	You activate or deactivate the feed-rate limitation for functional safety (FS). Only on machines with functional safety (FS). **Further information:** "Feed-rate limiting with functional safety (FS)", Page 486
AFC	You activate or deactivate Adaptive Feed Control (AFC, option 45). **Further information:** "AFC toggle switch in Program Run operating mode", Page 257
AFC settings	The control opens a selection menu with the following tables for AFC (option 45): ■ **AFC.TAB** for AFC basic settings ■ **AFC.DEP** settings file for teach-in cuts of the active NC program ■ **AFC2.DEP** log file of the active NC program **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 252
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC, option 145). **Further information:** "Active Chatter Control (ACC, option 145)", Page 260
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 372

Icon or button	Meaning
Breakpoints	When you select this button, the control opens the **Breakpoints** window with the following selection possibilities: ■ **Feed FMAX** You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit FMAX", Page 372 ■ **Skip block** If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** User's Manual for Programming and Testing ■ **Pause at M1** If this toggle switch is active, the control pauses the program run at each NC block with **M1**. If this toggle switch is inactive, the control grays out the **M1** syntax element. **Further information:** User's Manual for Programming and Testing
Skip block	If this toggle switch is active, the control ignores NC blocks hidden with a **/** character. If the toggle switch is active, the control grays out the NC blocks to be skipped. **Further information:** User's Manual for Programming and Testing
Pause at M1	If this toggle switch is active, the control pauses the program run at each NC block with **M1**. If this toggle switch is inactive, the control grays out the **M1** syntax element. **Further information:** User's Manual for Programming and Testing
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** User's Manual for Programming and Testing
Manual traverse	During a program run interruption you can move the axes manually. If **Manual traverse** is active, then the operating mode's icon in the control bar changes. **Further information:** "Manual traverse during an interruption", Page 378
Edit	If this toggle switch is active, then you can edit the pallet table. Active only if a pallet table is open **Further information:** User's Manual for Programming and Testing
3D ROT	During a program run interruption you can move the axes manually in the tilted working plane (option 8). **Further information:** "Manual traverse during an interruption", Page 378
Approach position	Return to contour after manual traverse of the machine axes during an interruption **Further information:** "Returning to the contour", Page 386
Block scan	The **Block scan** function allows you to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with **M3**. **Further information:** "Block scan for mid-program startup", Page 379

Icon or button	Meaning
Open in the editor	The control opens the active NC program and also called NC programs in the **Editor** operating mode. Active only if an NC program is open **Further information:** User's Manual for Programming and Testing
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Feed rate limit FMAX

The **FMAX** 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 **FMAX** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **FMAX** button in the function bar, the control opens the **Feed rate FMAX** window.

If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the control shows the feed rate in orange.

Further information: "Statusanzeigen", Page

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate FMAX** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M0**)
- Stop program run (e.g., with the **NC stop** key)
- Cancel the program run (e.g., with the **NC Stop** key in combination with 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 328

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview on the TNC bar", Page 115

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

- Selecting an operating mode
- Manual traverse of axes
- Checking Q parameters and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the setting for the programmed skipping of NC blocks with **/**

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the window **Q parameter list**
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

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

After an interruption of the program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 379

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 386

The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE

Danger of collision!

Program cancellation, manual intervention, or forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision.

- Rescind all programmed NC functions and transformations within the NC program
- Run a simulation before executing an NC program
- Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program
- Carefully verify the NC program in the **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 moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece reference point during a program run interruption, you must re-select the NC block to resume.

  Further information: "Block scan for mid-program startup", Page 379
- 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 **GS** workspace have an effect on program run, such as handwheel superimpositioning (option 44).

  Further information: "Global Program Settings (GPS, option 44)", Page 261

Definitions

Abbreviation	Definition
GS (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 displays a navigation path in the file information bar of the **Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when the program run is interrupted.

Related topics

- Program call
  Further information: User's Manual for Programming and Testing
- **Program** workspace
  Further information: User's Manual for Programming and Testing
- **Simulation** workspace
  Further information: User's Manual for Programming and Testing
- Interrupted program run
  Further information: "Interrupting, stopping or canceling program run", Page 373

Requirement

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

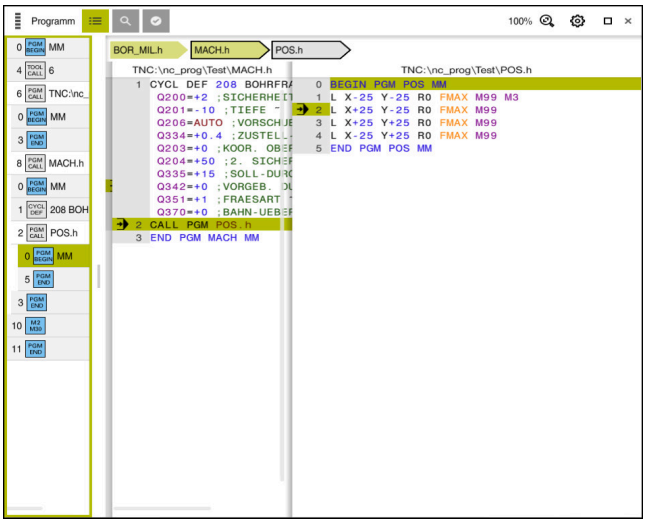
Description of function

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 layer 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.



NC programs called 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 structure items, including 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 position of the execution.

Further information: User's Manual for Programming and Testing

18.1.3 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

Use the **Tilt the working plane (3D ROT)** window to assign the reference system for traversing the axes (option 8).

Related topics

- Manual traverse of machine axes
  Further information: "Moving the machine axes", Page 147
- Manual tilting of the working plane (option 8)
  Further information: User's Manual for Programming and Testing

Description of function

When you select **Manual traverse**, you can move the axes with the axis keys of the control.

Further information: "Using axis keys to move the axes ", Page 148

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

Icon	Function	Meaning
	M-CS machine	Traversing in the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200
	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 204
	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 206
	**T-CS tool**	Traversing in the tool coordinate system **T-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 206

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

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

Notes

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). There is a risk of collision if the **3-D ROT** setting is incorrect!

- It is better to use the **T-CS** function
- Use a low feed rate

- On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.
  Refer to your machine manual.

18.1.4 Block scan for mid-program startup

Application

With the **BLOCK SCAN** function you can start an NC program at any desired NC block. The control will arithmetically account for workpiece machining up to this NC block. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs
  Further information: User's Manual for Programming and Testing
- Pallet tables and job lists
  Further information: User's Manual for Programming and Testing

Requirement

- The function must be enabled by your machine manufacturer.
  The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- **Internal stop** button
- Emergency stop
- Power failure

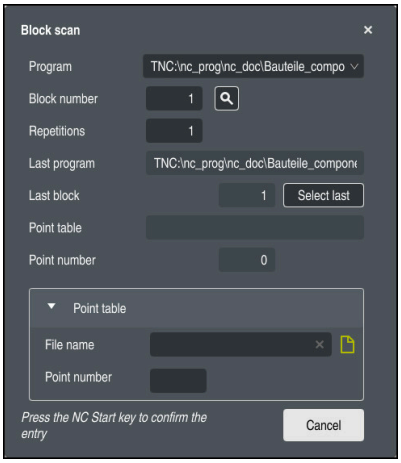
If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

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 382
- Multi-level block scan in subprograms and touch probe cycles
  Further information: "Performing a multi-level block scan", Page 383
- Block scan in a point table
  Further information: "Block scan in point tables", Page 384
- Block scan in pallet programs
  Further information: "Block scan in pallet tables", Page 385

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During a block scan you can activate or deactivate **Single Block** mode.

Block scan window



Block scan window with the interruption point saved and the **Point table** area opened

The **Block scan** window provides the following data:

Row	Meaning
Pallet number	Row number in the pallet table
Program	Path of the active NC program
Block number	Number of the NC block at which program run should start Use the **search** icon to select the NC block in the NC program.
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat.
Last pallet number	Pallet number that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last program	Path of the NC program that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last block	Number of the NC block that was active at the time of interruption Select the interruption point by using the **Select last** button.
Point file	Path of the point table In the **Point table** area
Point number	Row in the point table In the **Point table** area

Performing a single-level block scan

To start in an NC program by using a single-level block scan:

- Select the **Program Run** operating mode

- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

- If required, use **Select last** to start at a saved interruption point

- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

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

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 387

- 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 382

- Activate the **Single Block** switch as needed

- Press the **NC Start** key to execute individual NC blocks as needed

- Select **Continue block scan**
- Define the NC block for mid-program startup

- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

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

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 387

- 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.
- Select **Point table**
- > The control opens the **Point table** area.
- **Point file**: Enter the path of the point table
- **Point number**: Select the row number of the point table for mid-program startup

- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

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

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 387

> If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired point for mid-program startup in the **Point number** field. The first point in the point pattern has the number 0.
>
> **Further information:** User's Manual for Machining Cycles

Block scan in pallet tables

To start in a pallet table:

- Select the **Program Run** operating mode

- Select **Block scan**
- The control opens the **Block scan** window.
- **Pallet number**: Enter the row number of the pallet table
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

- If required, use **Select last** to start at a saved interruption point

- Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.

- Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.

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

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 387

> If the program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- Use the **Block scan** function at multiple levels

- The control only displays the dialogs required by the process in the pop-up window.
- A **Block scan** always takes place in a workpiece-oriented manner, even if you selected a tool-oriented machining method. After the block scan, the control continues working again in accordance with the selected machining method.

  Further information: User's Manual for Programming and Testing
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.

  Further information: "LBL tab", Page 121
- 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

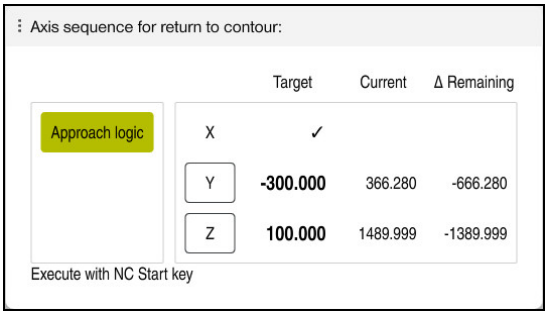
- Manual traverse during program run interruptions
  Further information: "Manual traverse during an interruption", Page 378
- **Block scan** function
  Further information: "Block scan for mid-program startup", Page 379

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.

Axis sequence for return to contour: window



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 387

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 388

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

Positioning manual axes

To position manual axes:

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

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

18.2 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: User's Manual for Programming and Testing
- Editing compensation tables in the NC program
  Further information: User's Manual for Programming and Testing
- Contents and creation of compensation tables
  Further information: User's Manual for Programming and Testing
- Contents and creation of a datum table
  Further information: User's Manual for Programming and Testing
- Activating a datum table in the NC program
  Further information: User's Manual for Programming and Testing

Description of function

The control opens the selected tables in **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 Program Run operating mode

To open the compensation tables from within **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 **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 test the NC program after changing the table values

- When you open a table in **Program Run** operating mode, the control will display the status **M** on the tab of the table. This status indicates that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

  Further information: "Status overview on the TNC bar", Page 115

18.3 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: "Retract workspace", Page 391
- **Positions**

  Further information: "Positions workspace", Page 109
- **Status**

  Further information: "Status workspace", Page 117

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
- **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 FMAX", Page 372

Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ **Machine axes**: Traverse in the machine coordinate system **M-CS** ■ **Tilted system**: Traverse in the working plane coordinate system **WPL-CS** (option 8) ■ **Tool axis**: Traverse in the tool coordinate system **T-CS** (option 8) ■ **Thread**: Traverse in the tool coordinate system **T-CS** with compensating movements of the spindle **Further information:** "Reference systems", Page 198
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes Effective for **Tilted system** traverse mode
Thread pitch	Thread pitch from the **PITCH** column of tool management Effective for **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ **Right-hand thread** ■ **Left-hand thread** Effective for **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective for **Tool axis** traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Note

NOTICE

Caution: Danger to the tool and workpiece!

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

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

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace

- ▶ Activate the **Retract** toggle switch

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

- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- ▶ Compare the displayed position values with the actual position values

- ▶ Select **OK**
- > The control closes the **Assume position values?** window
- ▶ Select the **Thread** traverse mode as needed
- ▶ Enter the thread pitch as needed
- ▶ Enter the direction of rotation as needed

- ▶ Select **Retract**
- ▶ Retract the tool with the axis keys or the handwheel

- ▶ Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.

- ▶ If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

19

Tables

19.1 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** and **Open File** workspaces.

In the **Quick selection** workspace, you can open some tables directly.

Further information: User's Manual for Programming and Testing

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: User's Manual for Programming and Testing

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table** and **Form** workspaces in every application.

Further information: "Table workspace", Page 396

Further information: "Form workspace for tables", Page 403

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

Further information: User's Manual for Programming and Testing

Buttons

The **Tables** operating mode provides the following buttons in the function bar:

Button	Meaning
Activate the preset	The control activates the currently selected row of the preset table as preset. **Further information:** "Preset table", Page 443
Undo	The control undoes the last change.
Redo	The control restores the change that has been undone.
GOTO record	The control opens the **GOTO jump instruction** window. The control jumps to the row number you have defined.
Edit	If the toggle switch is active, you can edit the table.
Insert tool	The control opens the **Insert tool** window that allows you to add a new tool to tool management. **Further information:** "Tool management ", Page 184 When you select the **Append** check box, the control inserts the tool below the last row of the table.
Insert line	The control inserts a row at the end of the table.
Reset row	The control resets all data contained in the row.
Delete tool	The control deletes the tool selected in the tool management. **Further information:** "Tool management ", Page 184
Delete row	The control deletes the currently selected row.
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 448
Mark row	The control marks the currently selected row.
Import	The control imports tool data. **Further information:** "Importing tool data", Page 186
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.

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

19.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Activate **Edit**

> The control enables the values for editing.

If the **Edit** toggle switch is active, you can edit the contents in both the **Table** workspace and the **Form** workspace.

Notes

- The control enables you to transfer tables from previous controls to the TNC7 and to adapt them automatically, if needed.
- If you open a table which has columns missing, the control will open the the **Incomplete table layout** window.

  In the **Incomplete table layout** window a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.
- If you, for example, have processed tables in a text editor, the control offers the **Update TAB / PGM** function. Use this function to complete an incorrect table format.

> Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., in the format).

19.2 Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.

Description of function

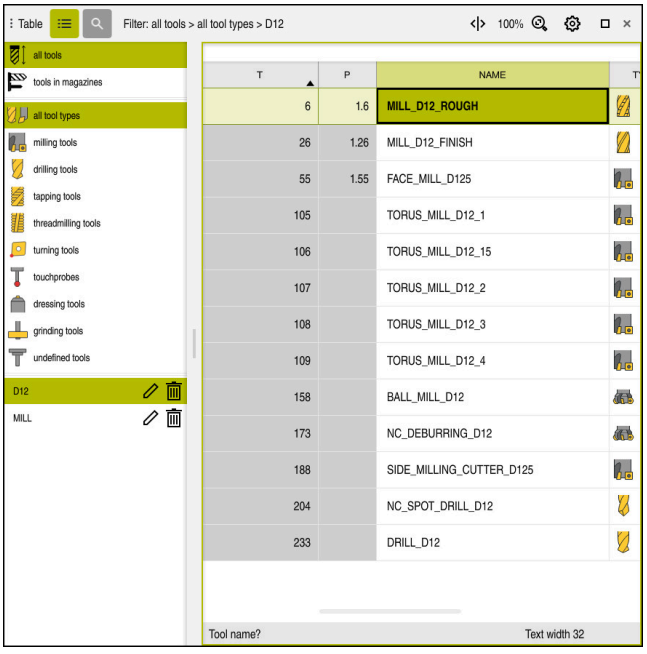


Table workspace

In **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column.

If the table allows it, you can also edit the table contents in this workspace.

Icons and shortcuts

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

Icon or shortcut	Function
	Open the filters **Further information:** "Filter column in the Table workspace", Page 397
	Open the search function **Further information:** "Search column in the Table workspace", Page 400
	Change column width **Further information:** "Changing the column width in the Table workspace", Page 402
100%	Font size of the table If you select the percent value, the control displays symbols for increasing and decreasing the font size.
	Set the font size of the table to 100%
	Open the settings in the **Tables** window **Further information:** "Settings in the Table workspace", Page 400
CTRL+A	Mark all rows
CTRL+BLANK	Mark the active row or end the marking function
SHIFT+↑	Additionally mark the row above
SHIFT+↓	Additionally mark the row below

Filter column in the Table workspace

You can filter the following table types:

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

Filtering in Tool management

The control provides the following default filters in the **Tool management**:

- **All tools**
- **Magazine tools**

According to the selection of **All tools** or **Magazine tools**, the control additionally offers the following default filters in the filter column:

- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Lathe tools**
- **Touch probes**
- **Dressing tools**
- **Grinding tools**
- **Undefined tools**

To display specific tool types, you must activate the desired filter or filters and deactivate the **All types** filter.

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- **all pockets**
- **spindle**
- **main magazine**
- **empty pockets**
- **occupied pockets**

Filtering in the Presets tablePresets

The control provides the following default filters in the **Presets** table:

- **Base transformations**
- **Offsets**
- **SHOW ALL**

User-defined filters

You can additionally create user-defined filters.

The control provides the following icons for each user-defined filter:

Icon	Meaning
	If you click on **Editing**, the control opens the **Search** column. You can edit and save the selected filter or save a filter under a new name. **Further information:** "Search column in the Table workspace", Page 400
	You can delete the selected filter.

To deactivate the user-defined filter, you need to activate the **All** filter and then to deactivate the user-defined filter.

Refer to your machine manual.

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

Logical connective operations between requirements and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter

  You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.
- OR operation between filters of the same type

  When you activate the default filters **Milling cutters** and **Lathe tools**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the requirements. The table row must contain either a milling cutter or a turning tool.
- AND operation between filters of different types

  You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

Search column in the Table workspace

You can search the following table types:

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

You can define multiple search conditions in the search function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
  Use the **Search in** selection menu to select the column.
- Perhaps an operator, such as **Contains** or **Equal to (=)**
  Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

> If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use **Add** to add several conditions. The conditions will have a combined effect when you perform the search. You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a filter. You can assign any name to the filter.

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

Settings in the Table workspace

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

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

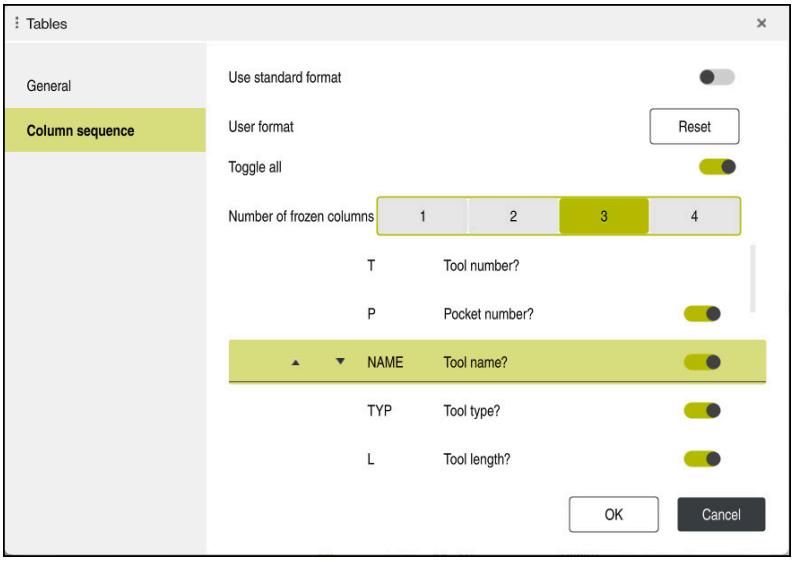
- **General**
- **Column sequence**

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.

Column sequence area



Tables window

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

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

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

19.2.1 Changing the column width in the Table workspace

To change the column width:

- Select the table column

- Select **Change column width**

> The control displays an arrow on the left and right in the header of the selected table column.

- Drag the arrow to the left or right

> The control reduces or enlarges the table column.

- Select other table columns if necessary

> If you select a further table column, then you need to select **Change column width** again.

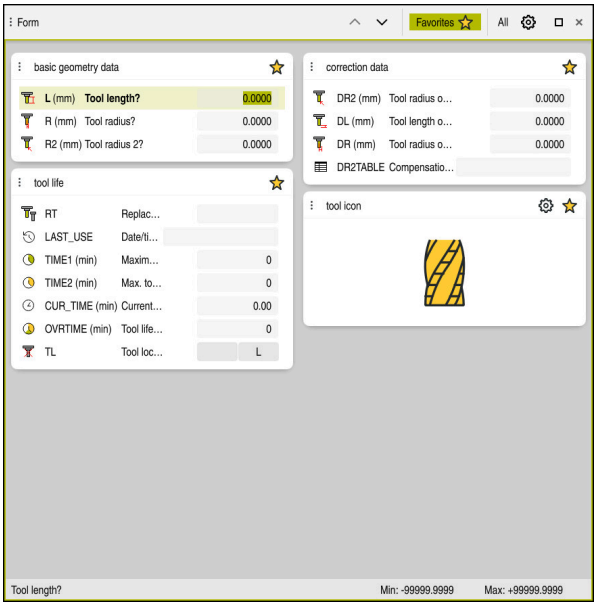
> You can also change the column width of non-editable table columns.

19.3 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



Form workspace in the **Favorites** view

The control displays the following information for each column:

- Icon of the column as needed
- Name of the column
- Unit of measure as needed
- Column description
- Current value

The control displays an icon of the selected tool type in the **Tool Icon** area. For the turning tools the icons also take into account the tool orientation and show where the relevant tool data will apply.

Further information: "Tool types", Page 168

If an input is invalid, the control displays an icon ahead of the input field. When you tap this icon, the control shows the cause of the error (e.g., **Too many characters**).

The control displays the contents of specific tables in groups within the **Form** workspace. In the **All** view, the control shows all groups. Use the **Favorites** function to select individual groups in order to configure a customized view. Use the gripper to arrange the groups.

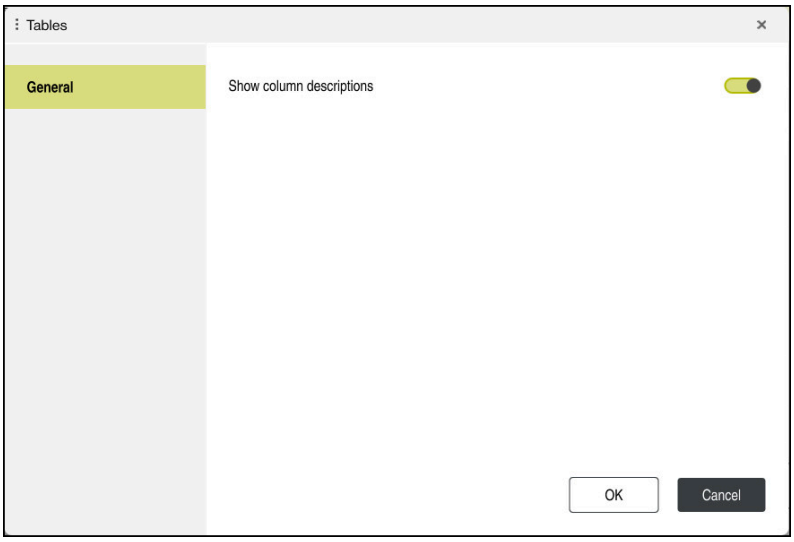
Icons

The **Table** workspace provides the following icons:

Icon or shortcut	Function
SHIFT+↑ SHIFT+↓	Navigate between table rows
	■ Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 404 ■ Change the size of the graphic in the **Tool Icon** area The control opens a selection window with the following settings: ■ **Small** ■ **Medium** ■ **Large**
	Favorite

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the column descriptions. The selected setting is modally effective.



19.4 Tool tables

19.4.1 Overview

This chapter describes the tool tables of the control.

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 405
- Turning tool table **toolturn.trn** (option 50)
  Further information: "Turning tool table toolturn.trn (option 50)", Page 415
- Grinding tool table **toolgrind.grd** (option 156)
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 419
- Dressing tool table **tooldress.drs** (option 156)
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 428
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 431

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 184

19.4.2 Tool table tool.t

Application

The tool table **tool.t** contains the data specific to drilling and milling tools. The tool table also contains all tool data that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 184
- Tool data required for milling or drilling tools
  Further information: "Tool data for milling and drilling tools", Page 172

Description of function

The file name of the tool table is **tool.t** and the tool table must be stored in the folder **TNC:\table**.

The **tool.t** tool table provides the following parameters:

Parameter	Meaning
T	**Tool number?** Row number in the tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 This parameter applies to all tools, regardless of technology. Input: **0.0...32767.9**

Parameter	Meaning
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 This parameter applies to all tools, regardless of technology. Input: **Text width 32**
L 	**Tool length?** Length of tool, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**
R 	**Tool radius?** Tool radius, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**
R2 	**Tool radius 2?** Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of, for example, ball-nose cutters or toroid cutters. **Further information:** User's Manual for Programming and Testing Input: **-99999.9999...+99999.9999**
DL 	**Tool length oversize?** Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter **L** Input: **-999.9999...+999.9999**
DR 	**Tool radius oversize?** Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter **R** Input: **-999.9999...+999.9999**
DR2 	**Tool radius oversize 2?** Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter **R2** Input: **-999.9999...+999.9999**

Parameter	Meaning
TL 	**Tool locked?** Tool is enabled or locked for machining: ■ No value entered: Enabled ■ **L**: Locked The control locks the tool after exceeding maximum tool age **TIME1**, maximum tool age 2 **TIME2** or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: No value, **L**
RT	**Replacement tool?** Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool at a suitable location. **Further information:** User's Manual for Programming and Testing 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 164 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **0.0...32767.9**
TIME1 	**Maximum tool age?** Maximum tool age in minutes If the current tool age **CUR_TIME** exceeds the **TIME1** value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**

Parameter	Meaning
TIME2 	**Max. tool age for TOOL CALL?** Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: ■ When the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool **RT** is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool **RT** at a suitable location. **Further information:** User's Manual for Programming and Testing The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**
CUR_TIME 	**Current tool age?** The current tool age equals the time during which the tool is cutting a workpiece. The 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. This parameter applies to all tools, regardless of technology. Input: **0...99999.99**
TYPE	**Tool type?** Depending on the selected tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Tool types", Page 168 **Further information:** "Tool management ", Page 184 This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **MILL**, **MILL_R**, **MILL_F**, **MILL_FACE**, **BALL**, **TORUS**, **MILL_CHAMFER**, **DRILL**, **TAP**, **CENT**, **TURN**, **TCHP**, **REAM**, **CSINK**, **TSINK BOR**, **BCKBOR**, **GF**, **GSF**, **EP**, **WSP**, **BGF**, **ZBGF**, **GRIND** and **DRESS**
DOC	**Tool description** This parameter applies to all tools, regardless of technology. Input: **Text width 32**
PLC	**PLC status?** Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology. Entry: **%00000000...%11111111**
LCUTS 	**Tooth length in the tool axis?** Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring. Input: **-99999.9999...+99999.9999**

Parameter	Meaning
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 of, for example, necks of end mills. Input: **0.0000...999.9999**
RN 	**Neck radius of the tool?** Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of e.g. neck of end mills or side milling cutters. Only if the useful length **LU** is longer than the **LCUTS** length of cutting edge, can the tool contain a neck radius **RN**. Input: **0.0000...999.9999**
ANGLE 	**Maximum plunge angle?** Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles. Input: **-360.00...+360.00**
CUT 	**Number of teeth?** Number of teeth of the tool for automatic tool measurement or cutting data calculation. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools **Further information:** User's Manual for Programming and Testing This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...99**
TMAT 	**Tool material?** Tool material from the tool material table **TMAT.tab** for cutting data calculation. **Further information:** User's Manual for Programming and Testing Selection by means of a selection window Input: **Text width 32**
CUTDATA 	**Cutting data table?** Select cutting data table with the ***.cut** or ***.cutd** file extensions for cutting data calculation. **Further information:** User's Manual for Programming and Testing Selection by means of a selection window Entry: **Text width 20**

Parameter	Meaning
LTOL 	**Wear tolerance: length?** Permitted tool length deviation in wear detection for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...5.0000**
RTOL 	**Wear tolerance: radius?** Permitted tool radius deviation in wear detection for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...5.0000**
R2TOL	**Wear tolerance: Radius 2?** Permitted tool radius 2 deviation in wear detection for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...9.9999**
DIRECT 	**Cutting direction?** Cutting direction of the tool for automatic tool measurement with a rotating tool: ■ **-**: **M3** ■ **+**: **M4** **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-**, **+**

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:** User's Manual for Measuring Cycles for Workpieces and Tools This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-99999.9999...+99999.9999**
L-OFFS 	**Tool offset: length?** Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools 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 (option 50) Input: **-99999.9999...+99999.9999**
LBREAK 	**Breakage tolerance: length?** Permitted tool length deviation in breakage detection for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...9.0000**
RBREAK 	**Breakage tolerance: radius?** Permitted tool radius deviation in breakage detection for automatic tool measurement. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...9.0000**
NMAX 	**Maximum speed [rpm]** Limitation of spindle speed for the programmed value including control by the potentiometer. Input: **0...999999**

Parameter	Meaning
LIFTOFF	**Lift-off allowed?** Automatic tool lift-off with active **M148** or allow **FUNCTION LIFTOFF**: ■ **Y**: activate **LIFTOFF** ■ **N**: deactivate **LIFTOFF** **Further information:** User's Manual for Programming and Testing Selection by means of a selection window Input: **Y**, **N**
TP_NO	**Number of the touch probe** Number of touch probe in touch probe table **tchprobe.tp** **Further information:** "Touch probe table tchprobe.tp", Page 431 Input: **0...99**
T-ANGLE 	**Point angle** Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of drills, for example. **Further information:** User's Manual for Machining Cycles Input: **-180...+180**
LAST_USE	**Date/time of last tool usage** Time of last tool presence in the spindle This parameter applies to all tools, regardless of technology. Input: **00:00:00 01.01.1971**...**23:59:59 31.12.2030**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table **Further information:** "Pocket table tool_p.tch", Page 435 Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
AFC	**Feedback-control strategy** Control setting for adaptive feed control AFC (option 45) from the **AFC.tab** table **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 252 Selection by means of a selection window Entry: **Text width 10**
ACC	**ACC active?** Activate or deactivate active chatter control ACC (option 145): ■ **Y**: activate ■ **N**: deactivate **Further information:** "Active Chatter Control (ACC, option 145)", Page 260 Selection by means of a selection window Input: **Y**, **N**
PITCH 	**Tool thread pitch?** Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. **Further information:** User's Manual for Machining Cycles Input: **-9.9999...+9.9999**

Parameter	Meaning
AFC-LOAD	**Reference power for AFC [%]** Tool-dependent reference power for AFC (option 45). 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 258 Input: **1.0...100.0**
AFC-OVLD1	**AFC overload warning level [%]** Cut-related tool wear monitoring for AFC (option 45). 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 259 Input: **0.0...100.0**
AFC-OVL2	**AFC ovrload switch-off level [%]** Cut-related tool load monitoring for AFC (option 45). 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 259 Input: **0.0...100.0**
KINEMATIC	**Tool-carrier kinematics** Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring. **Further information:** "Tool carrier management", Page 188 Selection by means of a selection window This parameter applies to all tools, regardless of technology. Entry: **Text width 20**
DR2TABLE	**Compensation val. table for DR2** Assigning a compensation value table ***.3dtc** for 3D tool radius compensation depending on the contact angle (option 92). This allows the control to compensate for inaccurate shapes of a ball-nose cutter or the deflection behavior of a touch probe, for example. **Further information:** User's Manual for Programming and Testing Selection by means of a selection window Entry: **Text width 16**
OVRTIME 	**Tool life expired** Time in minutes during which the tool may be used beyond the tool life defined in column **TIME2**. The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control uses the parameter when searching for tool names. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
RCUTS 	**Width of the indexable insert** Face-side width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., for indexable inserts). Input: **0...99999.9999**

Parameter	Meaning
DB_ID	**ID for central tool management** The database ID allows you to identify a tool (e.g., by using client applications within a tool management system). **Further information:** "Database ID", Page 164 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool. **Further information:** "Indexed tool", Page 164 This parameter applies to all tools, regardless of technology. Input: **Text width 40**
R_TIP	**Radius at the tip** Radius at the tool tip for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of countersinks, for example. Input: **0.0000...999.9999**

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 435
- 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 only changes the position of the tool in the simulation.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks 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 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.

19.4.3 Turning tool table toolturn.trn (option 50)

Application

The turning tool table **toolturn.trn** contains the data specific to turning tools.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 184
- Tool data required for turning tools
  Further information: "Tool data for turning tools (option 50)", Page 174
- Milling-turning operations on the control
  Further information: User's Manual for Programming and Testing
- General tool data, regardless of the technology
  Further information: "Tool table tool.t", Page 405

Requirements

- Combined milling-turning (software option 50)
- Turning tool is defined in **TYP** column of tool management
  Further information: "Tool types", Page 168

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 **toolturn.trn** turning tool table provides the following parameters:

Parameter	Meaning
T	Row number in the turning tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Tool name?** The tool name identifies a tool, for example when calling it. **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 Input: **Text width 32**
ZL 	**Tool length 1?** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2?** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**

Parameter	Meaning
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 159 Input: **-99999.9999...+99999.9999**
DZL 	**Oversize in tool length 1?** Delta value of tool length 1 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Oversize in tool length 2?** Delta value of tool length 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
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:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Cutting edge radius?** The control takes into account the cutter radius for tool tip radius compensation. **Further information:** User's Manual for Programming and Testing In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **TO**, **T-ANGLE**, and **P-ANGLE**. Input: **0...99999.9999**
DRS 	**Cutter radius oversize?** Delta value of cutter radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter **RS** Input: **-999.9999...+999.9999**

Parameter	Meaning
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:** User's Manual for Programming and Testing Refer to your machine manual. The control displays the tool orientations that are possible for each tool type. The machine manufacturer can change this assignment. In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **T-ANGLE**, and **P-ANGLE**. Input: **1...19**
SPB-INSERT 	**Angular offset?** Angular offset for recessing tools Input: **-90.0...+90.0**
ORI 	**Angle of spindle orientation?** Angle of tool spindle for aligning the turning tool Input: **-360.000...+360.000**
T-ANGLE 	**Tool angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **P-ANGLE**. Input: **0...179.999**
P-ANGLE 	**Point angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **T-ANGLE**. Input: **0...179.999**

Parameter	Meaning
CUTLENGTH  	**Cutting length of recessing tool** Length of the cutting edge of a turning or recessing tool The control monitors the length of the cutting edge in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the 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 Input: **0...99999.9999**
CUTWIDTH  	**Width of recessing tool** The control uses the width of a recessing tool for calculations within cycles. **Further information:** User's Manual for Machining Cycles Input: **0...99999.9999**
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:** User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter **CUTWIDTH** Input: **-99999.9999...+99999.9999**
TYPE 	**Type of turning tool** Depending on the selected turning tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the turning tools", Page 170 **Further information:** "Tool management ", Page 184 Selection by means of a selection window Input: **ROUGH**, **FINISH**, **THREAD**, **RECESS**, **BUTTON**, and **RECTURN**
WPL-DX-DIAM	**Compensation value for the workpiece diameter** Compensation value for the workpiece diameter with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 206 Input: **-99999.9999...+99999.9999**
WPL-DZL	**Compensation value for the workpiece length** Compensation value for the workpiece length with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 206 Input: **-99999.9999...+99999.9999**

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes 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 looks 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 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 435
- 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.

19.4.4 Grinding tool table toolgrind.grd (option 156)

Application

The grinding tool table **toolgrind.grd** contains the data specific to grinding tools.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 184
- Tool data required for grinding tools

  Further information: "Tool data for grinding tools (option 156)", Page 176
- Grinding operations on milling machines

  Further information: User's Manual for Programming and Testing
- Tool table for dressing tools

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 428
- General tool data, regardless of the technology

  Further information: "Tool table tool.t", Page 405

Requirements

- Jig grinding (software option 156)
- Grinding tool is defined in **TYPE** column of tool management
  Further information: "Tool types", Page 168

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

Further information: "Tool data for grinding tools (option 156)", Page 176

The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

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 identifying each tool unambiguously (e.g., for calling a tool). **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 The row number must match the tool number in the **tool.t** tool table Input: **0...32767**

Parameter	Meaning
NAME	**Name of grinding wheel** The tool name identifies a tool, for example when calling it. **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 Input: **Text width 32**
TYPE 	**Type of grinding wheel** Depending on the selected grinding tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the grinding tools", Page 170 **Further information:** "Tool management ", Page 184 Selection by means of a selection window Input: **GRIND_PIN**, **GRIND_CONE**, **GRIND_CUP**, **GRIND_CYLINDER**, **GRIND_ANGULAR** and **GRIND_FACE**
R-OVR 	**Radius of grinding wheel** Outermost radius of grinding tool After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
L-OVR 	**Overhang of grinding wheel** Length up to the outermost radius of the grinding tool, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
LO 	**Overall length** Absolute length of the grinding tool, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
LI 	**Length to the inner edge** Length up to the inner edge, with respect to the tool carrier reference point After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
B 	**Width** Width of the grinding tool After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**

Parameter	Meaning
G 	**Depth** Depth of grinding wheel After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.000000...999.999999**
ALPHA	**Angle for the slant** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.00000...90.00000**
GAMMA	**Angle for the corner** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **45.00000...180.00000**
RV 	**Radius at the edge for L-OVR** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
RV1 	**Radius at the edge for LO** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
RV2 	**Radius at the edge for LI** After initial dressing, you will no longer be allowed to edit this parameter. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
dR-OVR 	**Compensation of the radius** Delta value of the radius for tool compensation Is added to the parameter **R-OVR** Input: **-999.999999...+999.999999**
dL-OVR 	**Compensation of the overhang** Delta value of the overhang for tool compensation Is added to the parameter **L-OVR** Input: **-999.999999...+999.999999**
dLO 	**Compensation of the total length** Delta value of the total length for tool compensation Is added to the parameter **LO** Input: **-999.999999...+999.999999**
dLI 	**Compensation of the length to the inner edge** Delta value of the length up to the inner edge for tool compensation Is added to the parameter **LI** Input: **-999.999999...+999.999999**

Parameter	Meaning
R_SHAFT 	**Radius of the tool shank** Input: **0.00000...999.99999**
R_MIN 	**Min. permissible radius** If, after dressing, the actual radius is below the minimum permissible radius defined here, the control will display an error message. Input: **0.00000...999.99999**
B_MIN 	**Min. permissible width** If, after dressing, the actual width is below the minimum permissible width defined here, the control will display an error message. Input: **0.00000...999.99999**
V_MAX 	**Maximum permissible cutting speed** Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. Input: **0.000...999.999**
V	**Current cutting speed** Currently no function Input: **0.000...999.999**
W	**Tilt angle** Currently no function Input: **-90.00000...90.0000**
W_TYPE	**Tilted toward inner or outer edge** Currently no function Input: **-1**, **0**, **+1**
KIND	**Type of machining (internal/external grinding)** Currently no function Input: **0**, **1**
HW	**Wheel has a relief cut** Currently no function Input: **0**, **1**
HWA 	**Angle for relief cut on the outer edge** Input: **0.00000...45.00000**
HWI 	**Angle for relief cut on the inner edge** Input: **0.00000...45.00000**
INIT_D_OK	**Initial dressing performed** Initial dressing is the first dressing operation performed on the grinding wheel. Currently no function Input: **0**, **1**

Parameter	Meaning
INIT_D_PNR	**Dresser location for initial dressing** Dressing location used for initial dressing Input: **0...9999**
INIT_D_DNR	**Dresser number for initial dressing** Number of the dresser used for initial dressing Input: **0...32767**
MESS_OK	**Measure the grinding wheel** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Input: **0**, **1**
STATE	**Setup status** Currently no function Input: **%0000000000000000...%1111111111111111**
A_NR_D	**Dresser number (diameter dressing)** The control uses this parameter only if **Dressing tool with wear, COR_TYPE_DRESSTOOL** has been selected in parameter **COR_TYPE**. Tool number of the dresser being used Corresponds to the **T_DRESS** parameter in the tool management Input: **0...32767**
A_NR_A	**Dresser number (outer edge dressing)** Currently no function Input: **0...32767**
A_NR_I	**Dresser number (inner edge dressing)** Currently no function Input: **0...32767**
DRESS_N_D 012	**Dressing counter for diameter (specification)** Currently no function Input: **0...999**
DRESS_N_A 012	**Dressing counter for outer edge (specification)** Currently no function Input: **0...999**
DRESS_N_I 012	**Dressing counter for inner edge (specification)** Currently no function Input: **0...999**
DRESS_N_D_ACT 00Y	**Current dressing counter of the diameter** Currently no function Input: **0...999**
DRESS_N_A_ACT 00Y	**Current dressing counter of the outer edge** Currently no function Input: **0...999**

Parameter	Meaning
DRESS_N_I_ACT 	**Current dressing counter of the inner edge** Currently no function Input: **0...999**
AD 	**Retraction amount at the diameter** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
AA 	**Retraction amount at the outer edge** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
AI 	**Retraction amount at the inner edge** The control uses this parameter when using a cycle for dressing. **Further information:** User's Manual for Machining Cycles Input: **0.00000...999.99999**
FORM	**Wheel shape** Currently no function Input: **0.00...99.99**
A_PL	**Chamfer length at outside** Currently no function Input: **0.00000...999.99999**
A_PW	**Chamfer angle at outside** Currently no function Input: **0.00000...89.99999**
A_R1	**Corner radius at outside** Currently no function Input: **0.00000...999.99999**
A_L	**Length of outside** Currently no function Input: **0.00000...999.99999**
A_HL	**Length of relief cut, wheel depth at outside** Currently no function Input: **0.00000...999.99999**
A_HW	**Angle of relief cut at outside** Currently no function Input: **0.00000...45.00000**
A_S	**Side depth at outside** Currently no function Input: **0.00000...999.99999**
A_R2	**Angle of departure at outside** Currently no function Input: **0.00000...999.99999**

Parameter	Meaning
A_G	**Reserve at outside** Currently no function Input: **0.00000...999.99999**
I_PL	**Chamfer length at inside** Currently no function Input: **0.00000...999.99999**
I_PW	**Chamfer angle at inside** Currently no function Input: **0.00000...89.99999**
I_R1	**Corner radius at inside** Currently no function Input: **0.00000...999.99999**
I_L	**Length of inside** Currently no function Input: **0.00000...999.99999**
I_HL	**Length of relief cut, wheel depth at inside** Currently no function Input: **0.00000...999.99999**
I_HW	**Angle of relief cut at inside** Currently no function Input: **0.00000...45.00000**
I_S	**Side depth at inside** Currently no function Input: **0.00000...999.99999**
I_R2	**Angle of departure at inside** Currently no function Input: **0.00000...999.99999**
I_G	**Reserve at inside** Currently no function Input: **0.00000...999.99999**
COR_ANG	**Inclination angle of dressing tool** Currently no function Input: **0.00000**...**360.00000**
COR_TYPE	**Selection of compensation method** You can choose between the following compensation methods: ■ **Grinding wheel with compensation, COR_TYPE_GRINDTOOL** Compensation method removing material from the grinding tool **Further information:** User's Manual for Programming and Testing ■ **Dressing tool with wear, COR_TYPE_DRESSTOOL** Compensation method removing material from the dresser Selection by means of a selection window Input: **0**, **1**

Notes

- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with grinding tools.
- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- 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 looks 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 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 only changes 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 435

19.4.5 Dressing tool table tooldress.drs (option 156)

Application

The dressing tool table **tooldress.drs** contains the data specific to dressing tools.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 184
- Tool data required for dressing tools
  Further information: "Tool data for dressing tools (option 156)", Page 180
- Initial dressing
  Further information: User's Manual for Machining Cycles
- Grinding operations on milling machines
  Further information: User's Manual for Programming and Testing
- Tool table for grinding tools
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 419
- General tool data, regardless of the technology
  Further information: "Tool table tool.t", Page 405

Requirements

- Jig grinding (software option 156)
- Dressing tool is defined in **TYP** column of tool management
  Further information: "Tool types", Page 168

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**.

The **tooldress.drs** dressing tool table provides the following parameters:

Parameter	Meaning
T	Row number in the dressing tool table The tool number allows identifying each tool unambiguously (e.g., for calling a tool). **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Name of dressing tool** The tool name identifies a tool, for example when calling it. **Further information:** User's Manual for Programming and Testing You can define an index after the period. **Further information:** "Indexed tool", Page 164 Input: **Text width 32**
ZL 	**Tool length 1** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**

Parameter	Meaning
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 159 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 159 Input: **-99999.9999...+99999.9999**
DZL 	**Tool length oversize 1** Delta value of tool length 1 for tool compensation Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Tool length oversize 2** Delta value of tool length 2 for tool compensation Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
DYL 	**Tool length oversize 3** Delta value of tool length 3 for tool compensation Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Tool tip radius** Input: **0.0000...99999.9999**
DRS 	**Cutter radius oversize** Delta value of the cutter radius for tool compensation Is added to the parameter **RS** Input: **-999.9999...+999.9999**
TO 	**Tool orientation** The control uses the tool orientation to determine the position of the tool's cutting edge. Input: **1...9**
CUTWIDTH	**Width of tool (plate, roll)** Tool width of the tool types **dressing plate** and **dressing roll** Input: **0.0000...99999.9999**
TYPE 	**Type of dressing tool** Depending on the selected dressing tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the dressing tools", Page 170 **Further information:** "Tool management ", Page 184 Selection by means of a selection window Input: **DRESS_FIX_RADIUS**, **HORNED**, **DRESS_ROT_RADIUS**, **DRESS_FIX_FLAT** and **DRESS_ROT_FLAT**

Parameter	Meaning
N-DRESS	**Speed of the tool (dressing spindle)** Shaft speed of a dressing spindle or dressing roll Input: **0.0000...99999.9999**

Notes

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

  Further information: "Pocket table tool_p.tch", Page 435
- 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 looks 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 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 435

19.4.6 Touch probe table tchprobe.tp

Application

The touch probe table **tchprobe.tp** defines the touch probe and data for the probing process, such as the probing feed rate. If you use several touch probes, you can save separate data for each touch probe.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 184
- Touch probe functions

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 331
- Programmable touch probe cycles

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

Description of function

NOTICE

Danger of collision!

The control cannot use Dynamic Collision Monitoring (DCM) to protect L-shaped styluses against collisions. When using a touch probe with an L-shaped stylus there is a risk of collision!

- Carefully run in the NC program or program section in the **Program Run Single Block** operating mode
- Watch out for possible collisions!

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	**Sequential number of touch probe** You use this number to assign the touch probe to the data in the tool management column **TP_NO**. Input: **1...99**
TYPE 	**Selection of the touch probe?** The 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. Input: **TS120**, **TS220**, **TS249**, **TS260**, **TS440**, **TS444**, **TS460**, **TS630**, **TS632**, **TS640**, **TS642-3**, **TS642-6**, **TS649**, **TS740**, **TS 760**, **KT130**, **OEM**
CAL_OF1 	**TS center misalignmt. ref. axis? [mm]** According to the selection of the **STYLUS** column, this parameter has the following function: ■ **SIMPLE**: Offset of the touch probe axis to the spindle axis in the main axis ■ **L-TYPE**: Length of extension on an L-shaped stylus Input: **-99999.9999...+99999.9999**
CAL_OF2 	**TS center misalignmt. aux. axis? [mm]** Offset of the touch probe axis to the spindle axis in the secondary axis Input: **-99999.9999...+99999.9999**
CAL_ANG 	**Spindle angle for calibration?** According to the selection of the **STYLUS** column, this parameter has the following function: ■ **SIMPLE**: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible). ■ **L-TYPE**: The control orients the extension using the spindle angle. Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Input: **0.0000...359.9999**

Parameter	Meaning
F 	**Probing feed rate? [mm/min]** In the machine parameter **maxTouchFeed** (no. 122602), the machine manufacturer defines the maximum probing feed rate. If **F** is greater than the maximum probing feed rate, then the maximum probing feed rate will be used. Input: **0...9999**
FMAX 	**Rapid traverse in probing cycle? [mm/min]** Feed rate at which the control pre-positions the touch probe and positions it between the measuring points Input: **+10...+99999**
DIST 	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will display an error message. Input: **0.00100...99999.99999**
SET_UP 	**Set-up clearance? [mm]** Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value. Input: **0.00100...99999.99999**
F_PREPOS 	**Pre-position at rapid? ENT/NOENT** Speed for pre-positioning: ■ **FMAX_PROBE**: Pre-position at the speed from **FMAX** ■ **FMAX_MACHINE**: Pre-position at machine rapid traverse Input: **FMAX_PROBE**, **FMAX_MACHINE**
TRACK 	**Probe oriented? Yes=ENT/No=NOENT** Orienting the infrared touch probe in each probing process: ■ **ON**: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ **OFF**: The control will not orient the touch probe. If you change the **TRACK** parameter, you must recalibrate the touch probe. Input: **ON**, **OFF**
SERIAL 	**Serial number?** The control automatically edits this parameter of touch probes with an EnDat interface. Input: **Text width 15**
REACTION	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT** As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: ■ **NCSTOP**: Interrupt NC program ■ **EMERGSTOP**: Emergency stop, quick braking of the axes Input: **NCSTOP**, **EMERGSTOP**

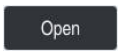
Parameter	Meaning
STYLUS	**Shape of the stylus** ■ **SIMPLE**: Straight stylus ■ **L-TYPE**: L-shaped stylus

Editing the touch probe table

To edit the touch probe table:

- ▶ Select the **Tables** operating mode

- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select the **tchprobe.tp** file in the **Open File** workspace



- ▶ Select **Open**
- > The control opens the **Touch probes** application.

- ▶ Activate **Edit**
- ▶ Select the desired value
- ▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

19.4.7 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**)

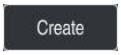
- Select the **Tables** operating mode

- Select **Add**

- Select **Create new table**
- > The control opens the **Create new table** window.
- Select a folder with a corresponding file extension (e.g., **t**)

- Select the desired prototype

- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter a name (e.g., **tool**)



- Select **Create**
- > The control opens the **Tool table** tab in **Tables** operating mode.
- Restart the control

- Acknowledge **Power interrupted** with the **CE** key

- Select the **Tool table** tab in **Tables** operating mode
- > The control uses the newly created table as a tool table.

19.5 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: User's Manual for Programming and Testing
- Tool table
  Further information: "Tool table tool.t", Page 405

Requirement

- The tool is defined in the tool management.
  Further information: "Tool management ", Page 184

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	**Pocket number?** Pocket number of the tool in the tool magazine Input: **0.0...99.9999**
T	**Tool number?** Row number of the tool from the tool table **Further information:** "Tool table tool.t", Page 405 Input: **1...99999**
TNAME	**Tool name?** Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. **Further information:** "Tool table tool.t", Page 405 Input: **Text width 32**
RSV	**Reserve pocket?** When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: - No value entered: Pocket is not reserved - **R**: Pocket is reserved Input: No value, **R**
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): - No value entered: No special tool - **S**: Special tool Input: No value, **S**
F	**Fixed pocket?** Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: - No value entered: No fixed pocket - **F**: Fixed pocket Input: No value, **F**

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 Input: No value, **L**
DOC	**Pocket comment?** The control automatically loads the tool comment from the tool table. **Further information:** "Tool table tool.t", Page 405 Input: **Text width 32**
PLC	**PLC status?** Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **%00000000...%11111111**
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **-99999.9999...+99999.9999**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **0...99**
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LAST_USE	**LAST_USE** The control automatically loads the date and time of the last tool call from the tool table. **Further information:** "Tool table tool.t", Page 405 Refer to your machine manual. Entry: **Text width 20**

Parameter	Meaning
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**

19.6 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 191
- Working with pallet tables
  Further information: User's Manual for Programming and Testing
- Tool data from the tool table
  Further information: "Tool table tool.t", Page 405

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
  Further information: "Creating the tool usage file", Page 192
- The **Generate tool-usage file** function setting is set to either **once** or **always**
  Further information: "Channel settings", Page 494

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file Input: **0...99999**
TOKEN	In the **TOKEN** column, the control uses one word to show which information is contained in the respective row: ■ **TOOL**: Data per tool call; listed in chronological order ■ **TTOTAL**: All data of a tool; listed in alphabetical order ■ **STOTAL**: Called NC programs; listed in chronological order ■ **TIMETOTAL**: Total tool usage time of an NC program ■ **TOOLFILE**: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table **tool.t** Input: **Text width 17**
TNR	Tool number If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767**
IDX	Tool index Input: **0...9**
NAME	Tool name Input: **Text width 32**
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Input: **0...9999999**
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece Input: **0...9999999**
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table Input: **-999999.9999...999999.9999**
BLOCK	NC block number of the tool call Input: **0...999999999**
PATH	Path of the NC program, the pallet table, or the tool table Input: **Text width 300**
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767.9**

Parameter	Meaning
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**. Input: **0...32767**
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**. Input: **-1...32767**
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed Input: **0**, **1**
LINENR	Row number of the pallet table in which the NC program is defined Input: **-1...99999**

Note

The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.

19.7 T usage order (option 93)

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

- Extended Tool Management (software option 93)
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 192

  Further information: "Tool usage file", Page 438

Description of function

When you select an NC program in **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 modify 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 164 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 164 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 435 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool management ", Page 184
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 164
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 438
TOOL TIME	Estimated time of tool change
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 438
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 435

19.8 Tooling list (option 93)

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

- Extended Tool Management (software option 93)
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 192

  Further information: "Tool usage file", Page 438

Description of function

When you select an NC program in **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 chronological order. You cannot modify 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 164 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 435 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 188
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 164
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 438
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 435

19.9 Preset table

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 154

Related topics

- Setting and activating presets

  Further information: "Preset management", Page 213

Description of function

The preset table is stored in the **TNC:\table** directory by default and is named **preset.pr**. In the **Tables** operating mode, the preset table is open by default.

Refer to your machine manual.

The machine manufacturer can define a different path for the preset table.

In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
	Active row
	Write-protected row

When you define a preset, the control opens a window with the following input options:

Icon or button	Function
	**actual position capture** The control opens or closes the position display of the status overview. When you select an axis, the control applies the selected value at **Enter new**. **Further information:** "actual position capture in the preset table", Page 448
Enter new	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 202 When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value. The entered value is active incrementally in the basic coordinate system **B-CS**.
Edit	The control accepts the entered value unchanged as table value. The entered value refers to the coordinate origin of the basic coordinate system **B-CS**.

Parameters of the preset table

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row Input: **0...99999999**
DOC	Comment Entry: **Text width 16**
X	X coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.99999...+99999.99999**
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.99999...+99999.99999**
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.99999...+99999.99999**
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.9999999...+99999.9999999**
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.9999999...+99999.9999999**
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 202 Input: **-99999.9999999...+99999.9999999**
X_OFFS	Position of the X axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.99999...+99999.99999**
Y_OFFS	Position of the Y axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.99999...+99999.99999**
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 200 Input: **-99999.99999...+99999.99999**

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 200 Input: **-99999.9999999...+99999.9999999**
B_OFFS	Axis angle of the B axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.9999999...+99999.9999999**
C_OFFS	Axis angle of the C axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.9999999...+99999.9999999**
U_OFFS	Position of the U axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.99999...+99999.99999**
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.99999...+99999.99999**
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 200 Input: **-99999.99999...+99999.99999**
ACTNO	Active workpiece preset The control automatically enters **1** in the active row. Input: **0**, **1**
LOCKED	Write-protection of the table row Entry: **Text width 16**

Refer to your machine manual.

In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Basic transformation and offset

The control interprets the basic transformations **SPA**, **SPB** and **SPC** as basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. During program execution, the control moves the linear axes in accordance with the basic rotation without any change in the workpiece position.

Further information: "Basic rotation and 3D basic rotation", Page 215

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.

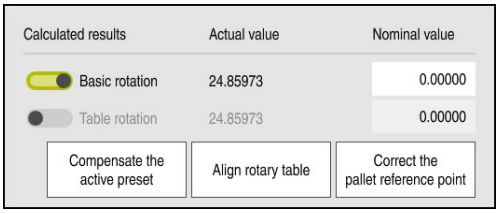
Further information: "Machine coordinate system M-CS", Page 200

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** probing function to determine the misalignment of a workpiece. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating the rotation of a workpiece", Page 343



Results of the **Rotation (ROT)** probing function**Rotation (ROT)**

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** into the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 449

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 449

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 locks the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

19.9.1 actual position capture in the preset table

To load the actual position of an axis into the preset table:

- Activate the **Edit** toggle switch
- Double-tap or double-click the table row to be changed (e.g., in the **X** column)

> The control opens a window with input options.

- Select **actual position capture**

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

- Select the desired value

> The control loads the value into the window and activates the **Enter new** button.

- Select **OK**

> The control calculates the table value that is needed and enters the value in the table.

- If required, close the position display of the status overview

19.9.2 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 locks 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.

19.9.3 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 locks 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.

19.9.4 Creating a preset table in inches

If you define inches as the unit of measure in the machine parameter **unitOfMeasure** (no. 101101), the unit of measure of the preset table will not be adjusted automatically.

To create a preset table in inches:

- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the **preset.pr** file (e.g., as **preset_mm.pr**)

- Select the **Tables** operating mode

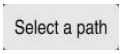
- Select **Add**

- Select **Create new table**

> The control opens the **Create new table** window.

- Select the **pr** folder

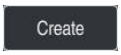
- Select the desired prototype



- Select **Select a path**

> The control opens the **Save as** window.

- Select the **table** folder
- Enter the name **preset.pr**



- Select **Create**

> The control opens the **Presets** tab in **Tables** operating mode.

- Restart the control

- Select the **Presets** tab in **Tables** operating mode

> The control uses the newly created table as a preset table.

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

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

- 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, you must reactivate the preset. Only then will the control accept the new value.

  Further information: "Activating presets", Page 214
- 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: User's Manual for Programming and Testing

Notes about machine parameters

- The machine manufacturer uses the optional machine parameter **initial** (no. 105603) to define a default value for each 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 in **Tables** operating mode.
- In the optional machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis how the control interprets offsets for the following NC functions:
  - **FUNCTION PARAXCOMP**
  - **FUNCTION POLARKIN** (option 8)
  - **FUNCTION TCPM** or **M128** (option 9)
  - **FACING HEAD POS** (option 50)

  Further information: User's Manual for Programming and Testing

19.10 Tables for AFC (option 45)

19.10.1 Basic AFC settings in AFC.tab

Application

In the **AFC.TAB** table, you can enter 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, option 45)", Page 252

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

Further information: "Description of function", Page 456

If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:

Parameter	Meaning
NR	Row number in the table Input: **0...9999**
AFC	Name of the control setting Enter this name in the **AFC** tool management column. It specifies the assignment of the control parameters to the tool. Input: Text width 10
FMIN	Feed rate at which the control will perform an overload response Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) 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 259 Input: **0...999**
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) 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 259 Input: **0...999**
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**

Parameter	Meaning
OVLD	Desired reaction of the control to overload: ■ **M**: Execution of a macro defined by the machine manufacturer ■ **S**: Immediate NC stop ■ **F:** NC stop once the tool has been retracted ■ **E**: Just display an error message on the screen ■ **L**: Disable active tool ■ **-**: No overload response If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum during that time, the control will conduct an overload response. In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options **M**, **E**, and **L**! Input: **M**, **S**, **F**, **E**, **L**, or **-**
POUT	Spindle power at which the control will detect that the tool exits the workpiece Enter the value in percent of the learned reference load Recommended input value: 8% In turning mode: Minimum load **Pmin** for tool monitoring (option 50) Input: **0...100**
SENS	Sensitivity (aggressiveness) of feedback control 50 is for slow feedback control, 200 for a very aggressive feedback control. An aggressive feedback control responds quickly and significantly changes the values, but it tends to overshoot. In turning mode: Activate the monitoring of the minimum load **Pmin** (option 50): ■ **1**: Evaluate **Pmin** ■ **0**: Do not evaluate **Pmin** Input: **0...999**
PLC	Value that the control will transfer to the PLC at the beginning of a machining step The machine manufacturer defines whether and which function will be performed by the control. Input: **0...999**

Creating an AFC.tab table

You need to create the table only if the table is missing in the **table** folder.

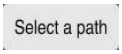
To create the **AFC.tab** table:

- Select the **Tables** operating mode

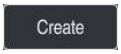
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.

- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **tab** folder

- Select the desired prototype



- Select **Select a path**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the desired name



- Select **Create**
- > The control opens the table.

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: User's Manual for Programming and Testing

19.10.2 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 451
- Setting up and using AFC

  Further information: "Adaptive Feed Control (AFC, option 45)", Page 252

Requirement

- Adaptive Feed Control (AFC, software option 45)

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 451

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)
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 will be recorded for this machining step. The control will overwrite any existing values in this line ■ **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 machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

19.10.3 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<name>.I.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, option 45)", Page 252

Requirement

- Adaptive Feed Control (AFC, software option 45)

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 conducted after the tool was retracted ■ **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 machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

19.10.4 Editing tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

To open a table for AFC:

- Select the **Program Run** operating mode

AFC settings

- Select **AFC settings**
- The control displays a selection menu. The control shows all existing tables 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

If you want to approach a position in the machine's working space while the guard door is open or if you execute a small infeed movement, you can use the electronic handwheel. The electronic handwheel allows you to traverse the axes and perform various functions provided by the control.

Related topics

- Incremental jog positioning
  Further information: "Incremental jog positioning of axes", Page 149
- Handwheel superimpositioning with GPS (option 44)
  Further information: "Function Handwheel superimp.", Page 269
- Handwheel superimpositioning with **M118**
  Further information: User's Manual for Programming and Testing
- Virtual tool axis **VT**
  Further information: User's Manual for Programming and Testing
- Touch probe functions in **Manual** operating mode
  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 331

Requirement

- Electronic handwheel (e.g., HR 550FS)
  The control supports the following electronic handwheels:
  - HR 410: Cable-bound handwheel without display
  - HR 420: Cable-bound handwheel with display
  - HR 510: Cable-bound handwheel without display
  - HR 520: Cable-bound handwheel with display
  - HR 550FS: Wireless handwheel with display, data transmission via radio

Description of function

You can use electronic handwheels in **Manual** or **Program Run** operating mode.

The HR 520 and HR 550FS portable handwheels feature a display that allows the control to show different types of information. You can use the handwheel soft keys for setup functions, such as the setting of presets or the activation of miscellaneous functions.

Once you have activated the handwheel with the handwheel activation key or the **Handwheel** toggle switch, you can operate the control only by using the handwheel. If you press the axis keys in this state, the control will display the message **Handwheel active: Handwheel-1, MB0**.

If more than one handwheel is connected to a control, you can activate or deactivate a handwheel only by pressing the handwheel activation key on the respective handwheel. You need to deactivate the active handwheel in order to be able to select another handwheel.

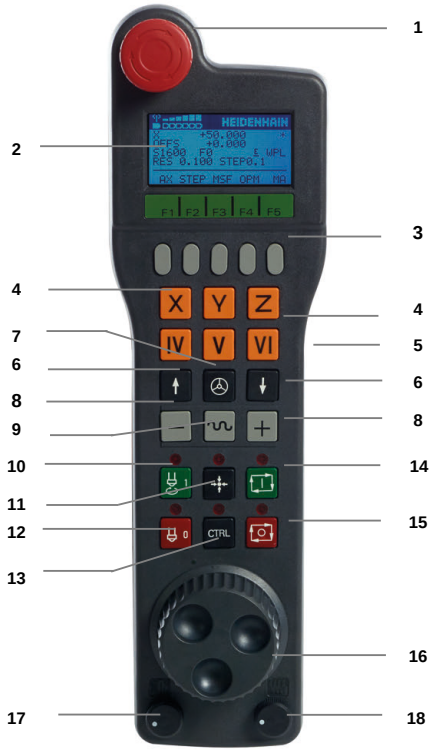
Functions in Program Run operating mode

You can perform the following functions in **Program Run** operating mode:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during an interruption of the program run (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys are used for operating.

  Further information: "Returning to the contour", Page 386
- Switch on/off the “Tilt working plane” function (handwheel soft keys **MOP** and then **3D**)

Operating elements of an electronic handwheel



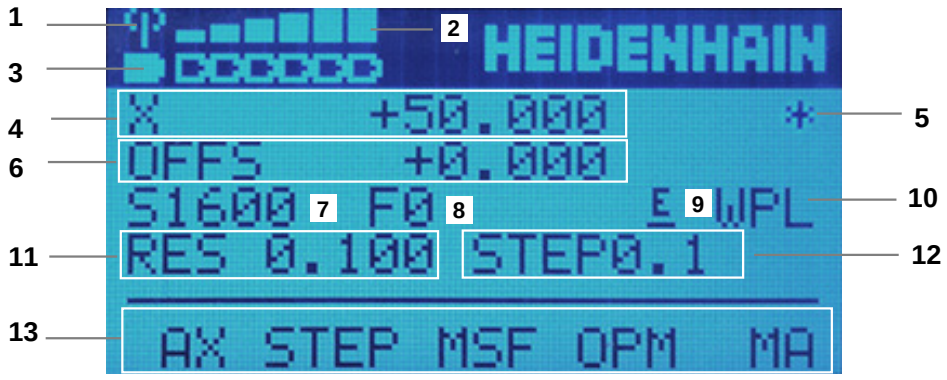
An electronic handwheel provides the following operating elements:

1 **EMERGENCY STOP** key
2 Handwheel display for status and for selecting functions
3 Handwheel soft keys
4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration
5 Permissive button

  The permissive button is on the rear side of the handwheel.
6 Arrow keys for defining the handwheel resolution
7 Handwheel activation key

8 Axis-direction key
Key for the direction of the traverse motion
9 Rapid traverse override for the traverse motion
10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)
11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)
12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)
13 **CTRL** key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)
14 **NC Start** key (machine-dependent function, key can be exchanged by the machine manufacturer)
15 **NC Stop** key
Machine-dependent function; key can be exchanged by the machine manufacturer
16 Handwheel
17 Spindle speed potentiometer
18 Feed rate potentiometer
19 Cable connection; not required for the HR 550FS wireless handwheel

Contents of an electronic handwheel display



The display of an electronic handwheel consists of the following areas:

1 Handwheel is in the docking station or radio mode is active
Only with HR 550FS wireless handwheel
2 Field strength
Six bars = maximum field strength
Only with HR 550FS wireless handwheel
3 Charging condition of battery
Six bars = maximum charging condition. A bar moves from the left to the right during recharging.
Only with HR 550FS wireless handwheel
4 **X+50.000**: Position of the selected axis

5 *****: Control in operation; program run has been started or axis is in motion

6 Handwheel superimpositioning from **M118** or the Global Program Settings GPS (option 44)

Further information: "Function Handwheel superimp.", Page 269

Further information: User's Manual for Programming and Testing

7 **S1000**: Current spindle speed

8 Feed rate at which the selected axis is moving

The control displays the current contouring feed rate while the program is running.

9 **E**: Error message

If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.

10 Active setting in the **3-D rotation** window:

- **VT**: **Tool axis** function
- **WP**: **Basic rotation** function
- **WPL**: **3D ROT** function

Further information: "3-D rotation window (option 8)", Page 220

11 Handwheel resolution

Distance that the selected axis moves per handwheel revolution

Further information: "Handwheel resolution", Page 464

12 Incremental jog active or inactive

If the function is active, the control will display the active traverse step.

13 Soft-key row

The soft key row provides the following functions:

- **AX**: Select the machine axis

  Further information: "Creating a positioning block", Page 466
- **STEP**: Incremental jog positioning

  Further information: "Incremental jog positioning", Page 466
- **MSF:** Execute various functions of the **Manual** operating mode (e.g., entering the feed rate **F**)

  Further information: "Entering miscellaneous functions M", Page 465
- **OPM**: Select the operating mode
  - **MAN**: **Manual** operating mode
  - **MDI**: **MDI** application in **Manual** operating mode
  - **RUN**: **Program Run** operating mode
  - **SGL**: **Single Block** mode of **Program Run** operating mode
- **MA**: Switching the magazine pockets

Handwheel resolution

The handwheel sensitivity specifies the distance an axis moves per handwheel revolution. The handwheel sensitivity results from the defined handwheel speed of the axis and the speed level used internally by the control. The speed level describes a percentage of the handwheel speed. The control calculates a specific handwheel sensitivity value for each speed level. The resulting handwheel sensitivity values are directly selectable with the handwheel arrow keys (only if incremental jog is not active).

The handwheel speed indicates the increment (e.g., 0.01 mm) traversed per handwheel detent position. You can change the handwheel speed with the handwheel's arrow keys.

If you have defined a handwheel speed of 1, the following handwheel resolutions are available:

Resulting handwheel sensitivity levels in mm/revolution and degrees/revolution:
0.0001/0.0002/0.0005/0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1

Resulting handwheel sensitivity levels in in/revolution:
0.000127/0.000254/0.000508/0.00127/0.00254/0.00508/0.0127/0.0254/0.0508/0.127/0.254/0.508

Examples for resulting handwheel sensitivity values:

Defined handwheel speed	Speed level	Resulting handwheel sensitivity
10	0.01%	0.001 mm/revolution
10	0.01%	0.001 degrees/revolution
10	0.0127%	0.00005 in/revolution

Effect of the feed-rate potentiometer when handwheel is active

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- Make sure to retract the tool before toggling between the handwheel and the machine operating panel.

The settings of the feed-rate potentiometer on the handwheel may differ from those on the machine operating panel. When you activate the handwheel, the control automatically activates the feed-rate potentiometer of the handwheel. When you deactivate the handwheel, the control automatically activates the feed-rate potentiometer of the machine operating panel.

In order to make sure that the feed rate does not increase while you switch over between the potentiometers, the feed rate is either frozen or reduced.

If the feed rate before switching over is higher than the feed rate after switching over, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching over is less than the feed rate after switching over, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because the activated feed-rate potentiometer will only then be effective.

20.1.1 Entering spindle speed S

To enter the spindle speed **S** by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F2** (**S**)
- Select the desired speed by pressing the **F1** or **F2** key
- Press the **NC Start** key

> The control activates the entered spindle speed.

> If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

20.1.2 Entering the feed rate F

To enter the feed rate **F** by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F3** (**F**)
- Select the desired feed rate by pressing the **F1** or **F2** key
- Load the new feed rate F with the **F3** (**OK**) handwheel soft key

> If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

20.1.3 Entering miscellaneous functions M

To enter a miscellaneous function by using an electronic handwheel:

- Press handwheel soft key **F3** (**MSF**)
- Press handwheel soft key **F1** (**M**)
- Select the desired M function number by pressing the **F1** or **F2** key
- Press the **NC Start** key

> The control activates the miscellaneous function

Further information: User's Manual for Programming and Testing

20.1.4 Creating a positioning block

Refer to your machine manual.

Your machine manufacturer can assign any function to the **Generate NC block** handwheel key.

To create a positioning block by using an electronic handwheel:

- Select the **Manual** operating mode
- Select the **MDI** application
- If necessary, select the NC block after which the positioning block should be inserted
- Activate the handwheel
- Press the **Generate NC block** key on the handwheel
- > The control inserts a straight line **L**, including all of the axis positions.

20.1.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a preset value.

To incrementally position an axis by using an electronic handwheel:

- Press the handwheel soft key F2 (**STEP**)
- Press the handwheel soft key 3 (**ON**)
- > The control activates incremental jog positioning.
- Set the desired jog increment by using the **F1** or **F2** keys

The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

- Confirm the selected jog increment by pressing the handwheel soft key F4 (**OK**)
- Use the **+** or **-** handwheel key to move the active handwheel axis in the corresponding direction
- > The control moves the active axis by the entered increment every time the handwheel key is pressed.

If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.

By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes

⚠ DANGER

Caution: hazard to the user!

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

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

NOTICE

Caution: Danger to the tool and workpiece!

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

- Place the handwheel in the handwheel holder when it is not in use
- Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
- Test the handwheel before machining

- The machine manufacturer can provide additional functions for the HR5xx handwheels.

  Refer to your machine manual.
- You can use the axis keys to activate the **X**, **Y**, and **Z** axes, as well as three other axes that can be defined by the machine manufacturer. Your machine manufacturer can also place the virtual axis **VT** on one of the free axis keys.

20.2 HR 550FS wireless handwheel

Application

With the HR 550FS wireless handwheel and its radio transmission characteristics, you can move farther away from the machine operating panel than with other handwheels. The HR 550FS wireless handwheel thus provides an important benefit, in particular for large machines.

Description of function

The HR 550FS wireless handwheel comes fitted with a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HRA 551FS handwheel holder and the HR 550FS handwheel together form one function unit.

HR 550FS handwheel

HRA 551FS handwheel holder

The HR 550FS handwheel can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you do not use the HR 550FS, always place it into the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.

Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.

If the control has triggered an emergency stop, you must reactivate the handwheel.

Further information: "Reactivating the handwheel", Page 472

If you happen to get close to the limit of the transmission range, the HR 550FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

Note

DANGER

Caution: hazard to the user!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- Check the radio connection of the handwheel for possible overlapping with other wireless devices
- Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

20.3 Configuration of wireless handwheel window

Application

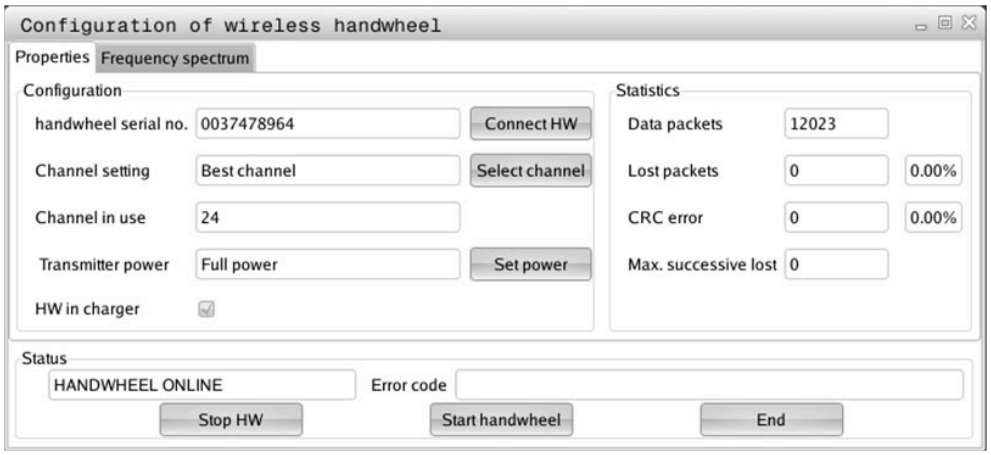
The **Configuration of wireless handwheel** window shows the connection data of the HR 550FS wireless handwheel and provides various functions for optimizing the radio connection, such as setting the radio channel.

Related topics

- Electronic handwheel
  Further information: "Electronic Handwheel", Page 459
- HR 550FS wireless handwheel
  Further information: "HR 550FS wireless handwheel", Page 468

Description of function

Use the **Set up wireless handwheel** menu item to open the **Configuration of wireless handwheel** window. The menu item is in the **Machine settings** group of the **Settings** application.



Areas of the Configuration of wireless handwheel window

Configuration area

In the **Configuration** area, the control displays different types of information about the connected wireless handwheel, such as the serial number.

Statistics area

In the **Statistics** area, the control displays information about the transmission quality.

If the received signal quality is impaired and no longer ensures a perfect, safe stop of the axes, the wireless handwheel will perform an emergency stop.

A high value under **Max. successive lost** is an indication of a limited quality of reception. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, there is a high risk of undesired disconnection.

If this occurs, try to improve the transmission quality by selecting a different channel or by increasing the transmitter power.

Further information: "Setting the radio channel", Page 472

Further information: "Selecting the transmission power", Page 471

Status area

In the **Status** area, the control displays the current status of the handwheel, such as **HANDWHEEL ONLINE** and pending error messages concerning the connected handwheel.

20.3.1 Assigning a handwheel to a handwheel holder

In order to assign a handwheel to a handwheel holder, the handwheel holder must be connected to the control hardware.

To assign a handwheel to a handwheel holder:

- Place the handwheel into the handwheel holder

- Select the **Home** operating mode

- Select the **Settings** application

- Select the **Machine settings** group

- Double-tap or double-click the **Set up wireless handwheel** menu item
- > The control opens the **Configuration of wireless handwheel** window.
- Select the **Connect HW** button
- > The control saves the serial number of the inserted wireless handwheel and displays it in the configuration window to the left of the **Connect HW** button.
- Select the **END** button
- > The control saves the configuration.

20.3.2 Selecting the transmission power

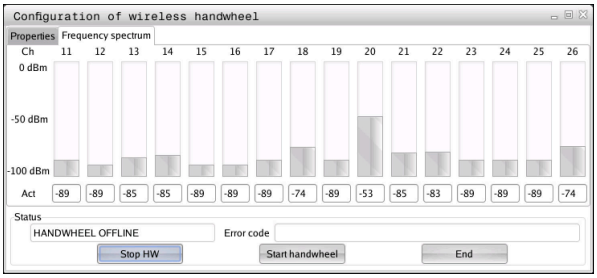
If you reduce the transmission power, the range of the wireless handwheel will decrease.

To set the transmission power of the handwheel:

- Open the **Configuration of wireless handwheel** window
- Select the **Set power** button
- > The control displays the three available power settings.
- Select the desired transmission power setting
- Select the **END** button
- > The control saves the configuration.

20.3.3 Setting the radio channel

If the wireless handwheel is started automatically, then the control tries to select the radio channel providing the best radio signal.



To set the radio channel manually:

- Open the **Configuration of wireless handwheel** window
- Select the **Frequency spectrum** tab
- Select the **Stop HW** button
- The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- Note the number of the channel with the least amount of radio traffic

> The smallest bar indicates the channel with the least amount of radio traffic.

- Select the **Start handwheel** button
- The control restores the connection to the wireless handwheel.
- Select the **Properties** tab
- Select the **Select channel** button
- The controls shows all available channel numbers.
- Select the number of the channel with the least amount of radio traffic
- Select the **END** button
- The control saves the configuration.

20.3.4 Reactivating the handwheel

To reactivate the handwheel:

- Open the **Configuration of wireless handwheel** window
- Use the **Start handwheel** button to reactivate the wireless handwheel
- Select the **END** button

21

Touch Probes

21.1 Setting up touch probes

Application

The **Device configuration** window allows you to create and manage all the workpiece and tool touch probes of the control.

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

Related topics

- Creating a workpiece touch probe with cable or infrared transmission by using the touch probe table

  Further information: "Touch probe table tchprobe.tp", Page 431
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)

  Further information: "Machine parameters", Page 542

Description of function

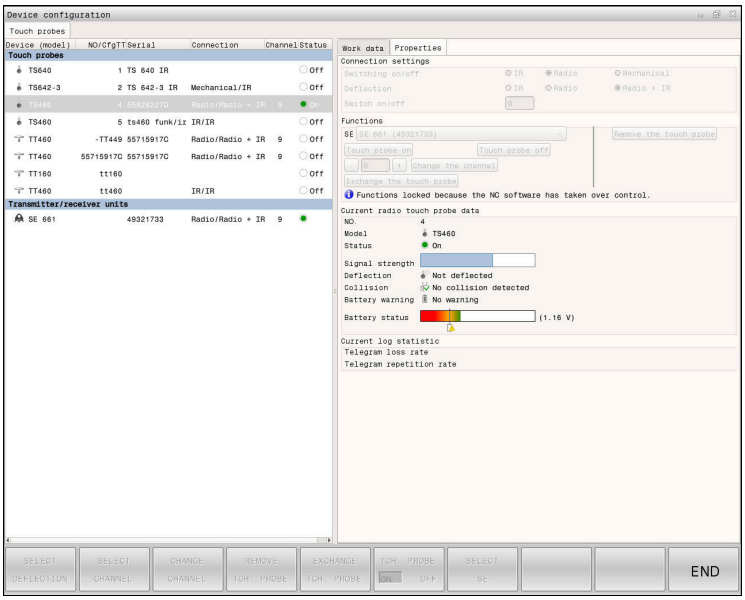
You open the **Device configuration** window in the **Machine settings** group of the **Settings** application. Double-tap or double-click the **Set up touch probes** menu item.

Further information: "Settings Application", Page 489

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

In order for the control to recognize the touch probe with radio transmission, you will require an **SE 661** transceiver with EnDat interface.

You define the new values in the **Work data** area.



Areas of the Device configuration window

Touch probes area

In the **Touch probes** area, the control displays all of the defined workpiece and tool touch probes, as well as the transceiver units. All other areas provide detailed information about the selected entry.

Work data area

For a workpiece touch probe, the control displays the values from the touch probe table in the **Work data** area.

For a tool touch probe, the control displays the values from the machine parameter **CfgTT** (no. 122700).

You can select and edit the displayed values. Under **Touch probes**, the control displays information about the active value (e.g., selection options). You can change the values of the tool touch probes only after entering the code number 123.

Properties area

In the **Properties** area, the control displays the connection data and diagnostic functions.

For touch probes with radio connection, the control displays the following information in **Current radio touch probe data**:

Display	Meaning
NO.	Number in the touch probe table
Type	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar graphic The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision recognized
Battery status	Display of the battery quality If the charge is less than the displayed bar, then the control outputs a warning.

The **Switching on/off** connection setting is preset based on the type of touch probe. Under **Deflection**, you can select how the touch probe is to transmit the signal when probing.

Deflection	Meaning
IR	Infrared probe signal
Radio	Radio probe signal
Radio + IR	The control selects the probe signal

If you activate the touch probe's radio connection by using the connection setting **Switch on/off**, then the signal will be retained even after a tool change. You need to use this connection setting to deactivate the radio connection.

Buttons

The control provides the following buttons:

Button	Function
CREATE TS ENTRY	Create a new workpiece touch probe You define the new values in the **Work data** area.
CREATE TT ENTRY	Create a new tool touch probe You define the new values in the **Work data** area.
SELECT DEFLECTION	Select the probe signal
SELECT CHANNEL	Select the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or wireless handwheels.
CHANGE CHANNEL	Change the radio channel
REMOVE TCH. PROBE	Delete the touch probe data The control deletes the entry from the **Device configuration** window and from the touch probe table or the machine parameters.
EXCHANGE TCH. PROBE	Save a new touch probe in the current row The control automatically overwrites the serial number of the replaced touch probe with the new number.
SELECT SE	Select the SE transceiver
SELECT IR POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT RADIO POWER	Select the strength of the radio signal You only need to change the signal strength if there is interference.

Note

In the machine parameter **CfgHardware** (no. 100102), the machine manufacturer defines whether the control will show or hide the touch probes in the **Device configuration** window. Refer to your machine manual.

22

Embedded Workspace and Extended Workspace

22.1 Embedded Workspace (option 133)

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 (option 133) to connect the Windows PC.

Related topics

- Remote Desktop Manager (option 133)
  Further information: "Remote Desktop Manager window (option 133)", Page 527
- Using Extended Workspace to operate a Windows PC through an additional connected monitor
  Further information: "Extended Workspace", Page 480

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (option 133)
- 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**.

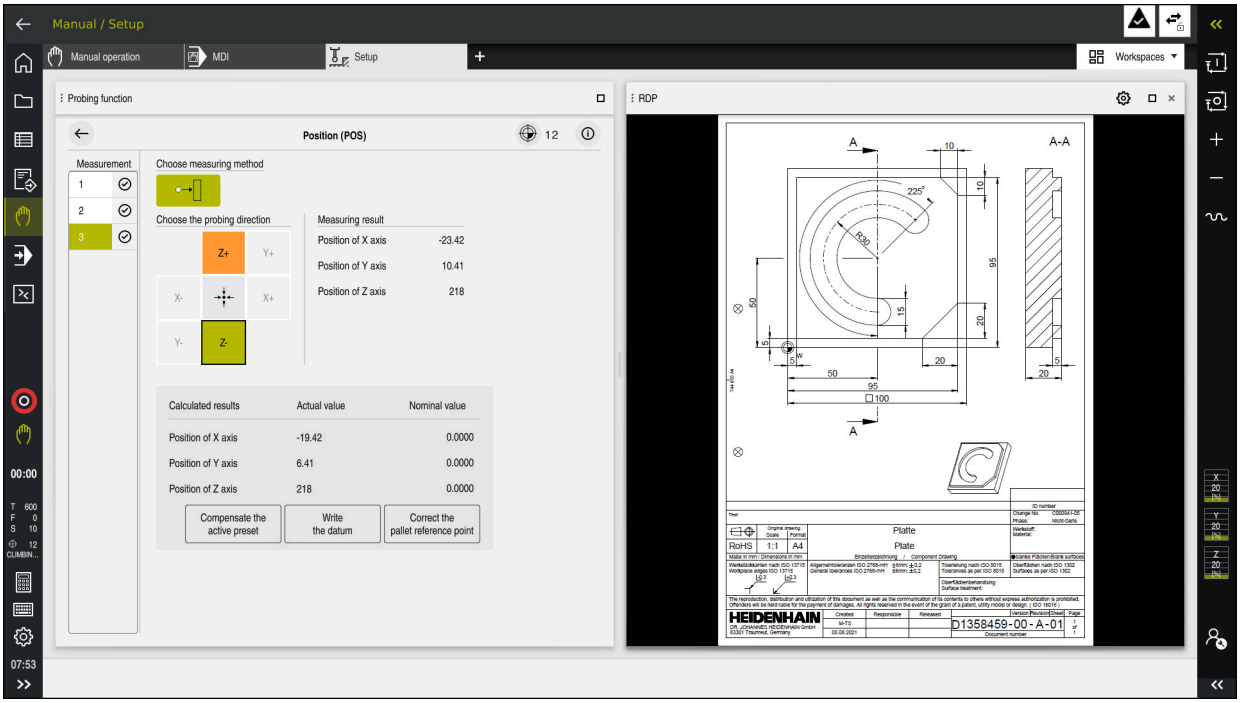
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 528

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 81



Embedded Workspace as workspace with opened PDF file

RDP settings window

If Embedded Workspace is open as a workspace, you can open the **RDP settings** window.

The **RDP settings** window contains the following buttons:

Button	Meaning
Reconnect	If the control could not establish a connection to the Windows PC, for example due to a timeout, press this button to try again. The control can also display this button in the operating mode and the workspace.
Adjust resolution	With this button the control rescales the user interface of the Windows PC to the size of the workspace.

22.2 Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

- Using Embedded Workspace to operate a Windows PC within the control's user interface (option 133

  Further information: "Embedded Workspace (option 133)", Page 478
- ITC hardware expansion

  Further information: "Hardware enhancements", Page 76

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 574
- Displaying and operating computers connected through Remote Desktop Managers (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 527

23

Integrated Functional Safety (FS)

Application

The safety concept of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety concept automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety concept.

Requirements

- Integrated functional safety (FS, basic version; software option 160) or Integrated functional safety (FS, full version; software option 161)
- Software options 162 to 166 or software option 169, if necessary
  Whether you need these software options depends on the machine's number of drives.
- The machine manufacturer must adapt the FS safety concept to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g. guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "Axis status menu item", Page 485

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of drives using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely Limited Speed	Safely limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:

Refer to your machine manual.

The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM 1	Operating mode **SOM_1**	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode **SOM_2**	Safe operating mode 2: Set-up mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified operators
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

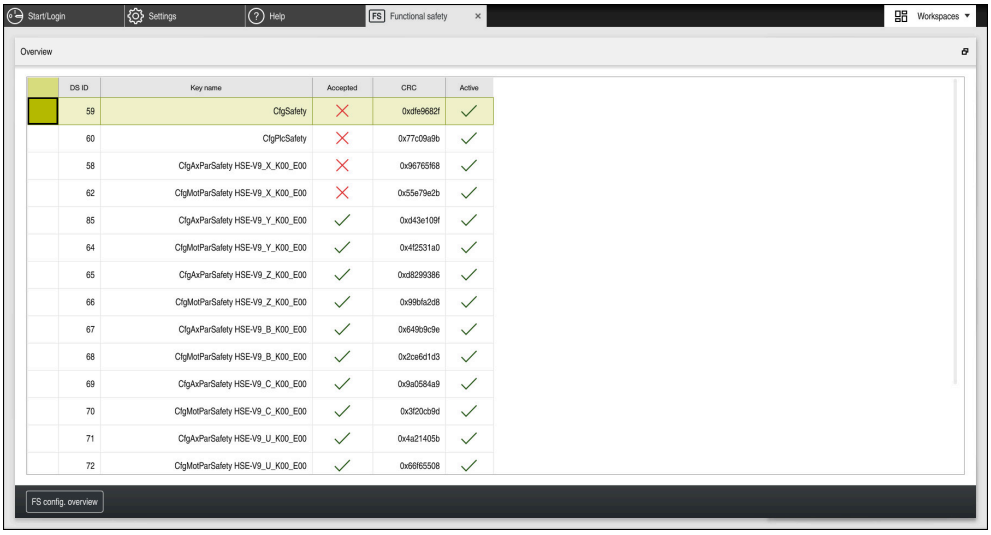
On a control with functional safety (FS), the control displays the monitored operating statuses of the speed **S** and feed rate **F** in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 110

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.

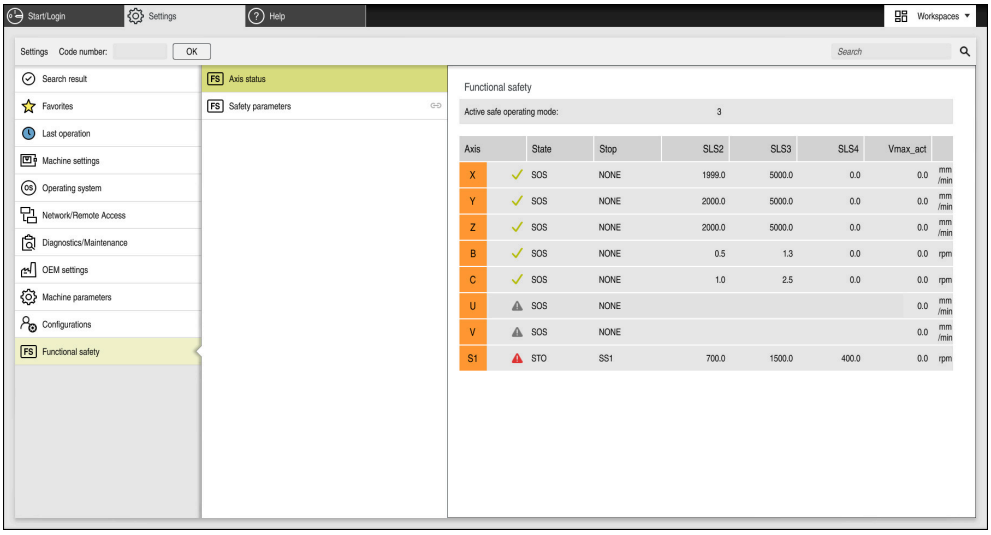


Functional safety application

Axis status menu item

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction **Further information:** "Functional safety (FS) in the Positions workspace", Page 483
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**.



Axis status menu item in the **Settings** application

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 487

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 487
⚠	The axis is not monitored by functional safety (FS) or is not configured as a safe axis.

Feed-rate limiting with functional safety (FS)

Refer to your machine manual.
This function must be adapted by your machine manufacturer.

With the **F limited** toggle switch you can prevent the SS1 reaction for safe stopping of drives when the guard door is opened.

With the **F limited** toggle switch the control limits the speed of the axes and rotational speed of the spindle to the values defined by the machine manufacturer. The limitation depends on the active safety-related SOM_x operating mode. You can select the safety-related operating mode with the keylock switch.

In the safety-related operating mode SOM_1, the control stops the axes and spindles when the guard door is opened.

In the **Positions** and **Status** workspaces the control shows the feed rate in orange.

Further information: "POS tab", Page 124

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 **Approach test position**
- > The control displays the axes that have not been tested in the **Positions** workspace.
- Select the desired axis in the **Positions** workspace
- Press the **NC start** key
- > The axis moves to the test position.
- > After the test position has been reached, the control issues a message.
- Press the **permissive button** on the machine operating panel
- > The control displays the axis as a tested axis.

NOTICE

Danger of collision!

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

- If necessary, move to a safe position before approaching the test positions
- Watch out for possible collisions

Notes

- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- 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 e.g. allow switching-on of the spindle and thus enable scratching of the workpiece while the guard door is open. Refer to your machine manual.

24

Settings Application

24.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Menu item
	Machine settings	■ **Machine settings** **Further information:** "Machine settings menu item", Page 493 ■ **General information** **Further information:** "General information menu item", Page 496 ■ **SIK** **Further information:** "SIK menu item", Page 497 ■ **Machine times** **Further information:** "Machine times menu item", Page 499 ■ **Set up touch probes** **Further information:** "Setting up touch probes", Page 474 ■ **Set up wireless handwheel** **Further information:** "HR 550FS wireless handwheel", Page 468
	Operating system	■ **Date/Time** **Further information:** "Adjust system time window", Page 500 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 501 ■ **About HEROS** **Further information:** "Information on licensing and use", Page 71 ■ **SELinux** **Further information:** "SELinux security software", Page 502 ■ **UserAdmin** **Further information:** "User management window", Page 558 ■ **Current User** **Further information:** "Active user window", Page 558 ■ **Touchscreen configuration** You can choose the touchscreen's sensitivity and whether touchpoints should be shown or hidden.

Icon	Category	Menu item
	Network/Remote Access	■ **Shares** **Further information:** "Network drives on the control", Page 503 ■ **Network** **Further information:** "Ethernet interface", Page 506 ■ **PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "OPC UA NC Server (options 56 to 61)", Page 513 ■ **OPC UA** **Further information:** "OPC UA NC Server (options 56 to 61)", Page 513 ■ **DNC** **Further information:** "DNC menu item", Page 518 ■ **Embedded Workspace** Show the connection status **Further information:** "Embedded Workspace (option 133)", Page 478 ■ **Printer** **Further information:** "Printers", Page 520 ■ **VNC** **Further information:** "VNC menu item", Page 523 ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 527 ■ **Real VNC Viewer** Define settings for external software accessing the control (e.g., for maintenance purposes); for network specialists ■ **Firewall** **Further information:** "Firewall", Page 533

Icon	Category	Menu item
	Diagnostics/Maintenance	■ **Terminal program** Enter and execute console commands ■ **HeLogging** Define settings for internal diagnostic files ■ **Portscan** **Further information:** "Portscan", Page 536 ■ **perf2** Check processor load and process load ■ **RemoteService** **Further information:** "Remote servicing", Page 537 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 538 ■ **TNCdiag** **Further information:** "TNCdiag", Page 542 ■ **TNCscope** Software for data recording ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 538 ■ **Touchscreen cleaning** The control disables the touchscreen for 90 seconds. ■ **Update documentation** **Further information:** "Update documentation", Page 540
	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 542
	Parameter files	Settings for the machine manufacturer
	Configurations	**Configurations** **Further information:** "Configuring the control's user interface", Page 547
FS	**Functional safety**	■ **Axis status** **Further information:** "Axis status menu item", Page 485 ■ **Safety parameters** **Further information:** "Functional safety application", Page 484

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	Function
123	Editing machine-specific user parameters **Further information:** "Machine parameters", Page 542
555343	Special functions for programming with variables **Further information:** User's Manual for Programming and Testing
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 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: User's Manual for Programming and Testing

Description of function

Unit of measure area

In the **Unit of measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10 before entering it.

Channel settings

The control displays the channel settings separately for the **Editor** operating mode and the **Manual** and **Program Run** operating modes.

You can define the following settings:

Setting	Meaning
Active kinematics	Use the **Active kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, are 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 191 You select when the control should generate a tool-usage file: ■ **Never** The control does not generate a tool-usage file. ■ **Once** The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ **Always** When you simulate or run an NC program, the control will generate a tool-usage file each time.

Traverse limits

Use the **Traverse limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents **Valid** or **Invalid**.
Lower limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: User's Manual for Programming and Testing

Notes

NOTICE

Danger of collision!

You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!

- Use the **Active kinematics** function for the simulation only
- Use the **Active kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active kinematics** function can be selected.
- You can open the tool-usage file in the **Tables** operating mode.
  Further information: "Tool usage file", Page 438
- If the control has generated a tool-usage file for an NC program, then there are contents in the **T usage order** and **Tooling list** tables (option 93).
  Further information: "T usage order (option 93)", Page 440
  Further information: "Tooling list (option 93)", Page 442

24.4 General information menu item

Application

In the **General information** menu item of the **Settings** application, the control provides the information about the control and the machine.

Description of function

Version information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ **Control model** Designation of the control (managed by HEIDENHAIN) ■ **NC-SW** Number of the NC software (managed by HEIDENHAIN) ■ **NCK** Number of the NC software (managed by HEIDENHAIN)
PLC	**PLC-SW** Number or name of the PLC software (managed by the machine manufacturer)

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

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 514

Machine information area

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 514

24.5 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 64

Description of function

SIK information area

The control displays the following information:

- **Serial number**
- **Control model**
- **Performance class**
- **Features**
- **Status**

OEM key area

In the **OEM key** area the machine manufacturer can define a manufacturer-specific password for the control.

General key area

In the **General key** area the machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing).

The control indicates the status of the general key:

Status	Meaning
NONE	The general key has not yet been used for this software version.
dd.mm.yyyy	Date up to which all software options will be available. Once the general key has expired, it cannot be used again.
EXPIRED	The general key has expired for this software version.

If the software version of the control is increased (e.g., by an update), then the **General key** can be used again.

Software options area

In the **Software options** area the control shows all available software options in a table.

Column	Meaning
#	Number of the software option
Option	Name of the software option
Expiration date	The machine manufacturer can enable software options for a limited time. In this case the control shows in this column the date through which the software option is available.
	The machine manufacturer uses the **Set** button to enable a software option. For enabled software options the control displays the text **Enabled**.

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** at the end of the row.

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**.

24.6 Machine times menu item

Application

In the **Machine times** menu item of the **Settings** application, the control shows the run times since initial setup.

Related topics

- Date and time of the control
  Further information: "Adjust system time window", Page 500

Description of function

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 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 an NTP server synchronization.

Related topics

- Run times of the machine tool
  Further information: "Machine times menu item", Page 499

Description of function

The **Date/Time** menu item opens the **Adjust system time** window. The menu item is in the **Operating system** group of the **Settings** application.

The **Adjust system time** window consists of 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.

24.8 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control
  Further information: "Machine parameters", Page 542

Description of function

You can't define two different conversational languages for the operating system and control.

The **Language/Keyboards** menu item opens the **helocale** window. The menu item is in the **Operating system** group of the **Settings** application.

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

24.8.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

- Select the **Settings** application
- Enter the code number 123
- Select **OK**
- Select **Machine parameters**
- Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- Navigate to the machine parameter **ncLanguage** (no. 101301)
- Select the desired language

- 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.

Note

Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.

- **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

  Further information: "Changing the language", Page 502
- **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

24.9 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions, i.e. 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 533

Description of function

The **SELinux** menu item opens the **Security Policy Configuration** window. The menu item is in the **Operating system** group of the **Settings** application.

The access control of **SELinux** is regulated as follows by default:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Only explicitly selected programs can modify safety-relevant files, such as **SELinux** system files or HEROS boot files.
- 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** with the **Security Policy Configuration** window is usually password-protected by the machine manufacturer (refer to the relevant 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.

24.10 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management
  Further information: User's Manual for Programming and Testing
- Network settings
  Further information: "Ethernet interface", Page 506

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

The **Shares** menu item opens the **Mount Setup** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can also open the window with the **Mount network share** button of the **Files** operating mode.

Further information: User's Manual for Programming and Testing

You can define any number of network drives, but only seven can be connected at a time.

Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the **Mount** column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the **Auto** column if an active automatic connection exists.
Add	Define a new connection **Further information:** "Mount assistant window", Page 505
Remove	Delete an existing connection
Copy	Copy connection **Further information:** "Mount assistant window", Page 505
Edit	Edit the connection settings **Further information:** "Mount assistant window", Page 505
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.

Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ **Drive name:** Network drive name in the file management of the control The control permits only capitals, with a colon (**:**) at the end. ■ **Private network drive** When user administration is 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** When user administration is 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 506
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 write the domain into the user name, but as an option.
vers=2.1	Protocol version

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
sec=ntlm	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.
nfsvers=2	Protocol version

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

24.11 Ethernet interface

Application

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network.

Related topics

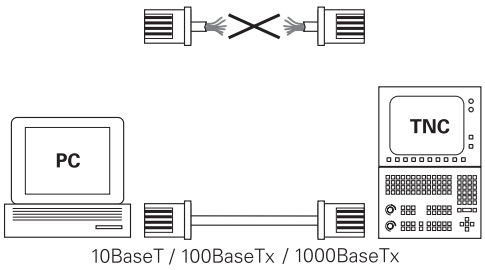
- Firewall settings
  Further information: "Firewall", Page 533
- Network drives on the control
  Further information: "Network drives on the control", Page 503
- External access
  Further information: "DNC menu item", Page 518

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



You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

Use a Twisted Pair cable to connect the control to your network.

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 578 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.

24.11.1 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 586
- Firewall settings

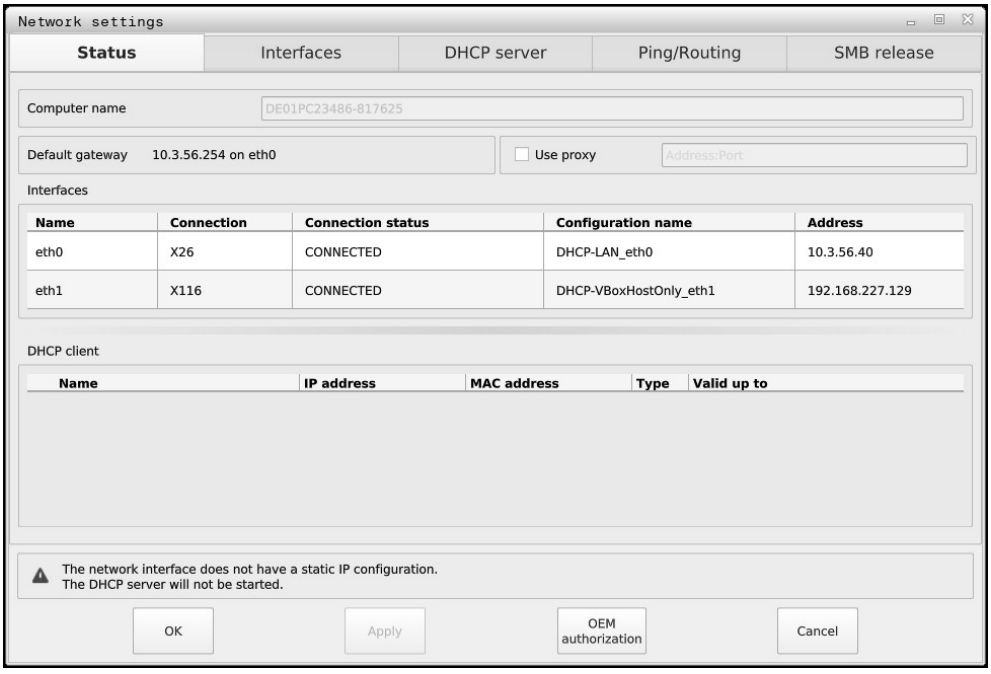
  Further information: "Firewall", Page 533
- Network drives on the control

  Further information: "Network drives on the control", Page 503

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Network



Network settings window

Status tab

The **Status** tab contains the following information and settings:

Domain	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name.
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the **address** and the **port** of a proxy server in the network.
Interfaces	The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty. The control displays the following information in the table: ■ **Name**, e.g. **eth0** ■ **Connection**, e.g. **X26** ■ **Connection status**, e.g. **CONNECTED** ■ **Configuration name**, e.g. **DHCP** ■ **Address**, e.g. **10.7.113.10** **Further information:** "Interfaces tab", Page 510
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 network 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.

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 586 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 586

The control additionally offers the following functions:

- **Set standard values**

  The control opens a pop-up window. You can import and activate profiles that you exported or that were already entered in the factory default setting.

  Further information: "Exporting and importing a network profile", Page 512
- **Configuration name**

  You can add, edit or remove profiles for the network connection.

  > 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.

  Further information: "Network configuration with Advanced Network Configuration", Page 586

DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).

Refer to your machine manual.

Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Domain	Information or Setting
Ping	**Address:Port** and **Address:** You can enter the IP address of the computer and possibly the port number for checking the network connection. Entry: Four numerical values separated by dots and, if necessary, a port number separated by a colon (e.g., **10.7.113.10:22**) As an alternative, you can enter the name of the computer whose connection you want to check. Starting and stopping the test ■ **Start** button: starts the test The control displays status information in the ping field. ■ **Stop** button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

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 desired network profile
- Press **OK**
- > The control saves the network profile in the **TNC:/etc/sysconfig/net** directory.

> You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:

- Open the **Network settings** window
- Select the **Interfaces** tab
- Select **Set standard values**
- > The control opens a window.
- Select **User**
- 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

Notes

- Preferably restart the control after making changes in the network settings.
- The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.

  Further information: "Conversational language of the control", Page 501

24.12 OPC UA NC Server (options 56 to 61)

24.12.1 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 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: "OPC UA connection wizard function (options 56 to 61)", Page 516

Requirements

- OPC UA NC Server (software options 56 to 61)
  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).
- Firewall configured
  Further information: "Firewall", Page 533
- OPC UA client supports the **Security Policy** and the authentication method of the **OPC UA NC Server**:
  - **Security Mode: SignAndEncrypt**
  - **Algorithm: Basic256Sha256**
  - **User Authentication: X509 certificates**

Description of function

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- Read and write tool data (the corresponding right is required)
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (the corresponding right is required)

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: "Machine information area", Page 496
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)
  Further information: "Info about machine manufacturer area", Page 496

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives.

The following actions are permitted:

- Creation and deletion of folders
- Reading, editing, copying, moving, creating, and deleting of files.

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. You can, for example, transfer automatically created service files at any time as long as the operating system is active.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- ▶ System-relevant files must be edited only by authorized specialists

Required certificates

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them, the application instance certificates, in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function.

All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- ▶ Open the **Current User** HEROS function
- ▶ Select **SSH keys and certificates**
- ▶ Press the **Import certificate** soft key
- > The control opens a pop-up window.
- ▶ Select the certificate
- ▶ Select **Open**
- > The control imports the certificate.
- ▶ Press the **Use for OPC UA** soft key

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
  - File format: *.der
  - Signature with hash SHA256
  - Validity period of at most 5 years is recommended
- Client certificates
  - Host name of the client
  - Application URI of the client
- Server certificates
  - Host name of the control
  - Application URI of the server according to the following structure: urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
  - Validity period of 20 years maximum

Note

OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.

24.12.2 OPC UA menu item (options 56 to 61)

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

Select the **OPC UA** menu item in the **Network/Remote Access** group.

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
OPC UA connection wizard	Open the **OPC UA NC Server connection wizard** window **Further information:** "OPC UA connection wizard function (options 56 to 61)", Page 516
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window **Further information:** "OPC UA license settings function (options 56 to 61)", Page 517
Host computer operation	Activate or deactivate host computer operation with a toggle switch **Further information:** "DNC area", Page 518

24.12.3 OPC UA connection wizard function (options 56 to 61)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection wizard** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning an OPC UA client to a software option 56 to 61 with the **OPC UA NC Server - License Settings** window
- Managing certificates with the **PKI Admin** menu

Description of function

The **OPC UA connection wizard** function of the **OPC UA** menu item opens the **OPC UA NC Server connection wizard** window.

Further information: "OPC UA menu item (options 56 to 61)", Page 516

The assistant features the following steps:

- Export **OPC UA NC Server** certificates
- Import the certificates of the OPC UA client application
- Assign each of the available **OPC UA NC Server** software options to an OPC UA client application
- Import user certificates
- Assign user certificates to users
- Configure the firewall

If at least one of the options 56 to 61 is active, then, when booting for the first time, the control creates the server certificate as part of a self-generated certificate chain. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. Please contact your IT department.

Note

The **OPC UA NC Server connection wizard** 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.

24.12.4 OPC UA license settings function (options 56 to 61)

Application

Use the **OPC UA NC Server - License Settings** window to assign one OPC UA client application to one of the software options 56 to 61.

Related topics

- Setting up an OPC UA client application with the **OPC UA connection wizard** function

  Further information: "OPC UA connection wizard function (options 56 to 61)", Page 516

Description of function

After using the **OPC UA connection wizard** 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 activate the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

24.13 DNC menu item

Application

With the **DNC** menu item you can grant or restrict access to the control (e.g., connections over a network).

Related topics

- Connecting network drives
  Further information: "Network drives on the control", Page 503
- Setting up a network
  Further information: "Ethernet interface", Page 506
- TNCremo
  Further information: "PC software for data transfer", Page 581
- Remote Desktop Manager (option 133)
  Further information: "Remote Desktop Manager window (option 133)", Page 527

Description of function

The **DNC** area contains the following icons:

Icon	Meaning
	External access to the control is active
	Add computer-specific connection
	Edit computer-specific connection
	Delete computer-specific connection

DNC area

In the **DNC** area you use toggle switches to activate the following functions:

Switches	Meaning
DNC access permitted	Permit or block all accesses to the control through a network or a serial connection
TNCopt full access allowed	Depending on the machine, permit or block access for diagnostics or initial setup software
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation If host computer operation is active, the control displays the **Host computer is active** message in the info bar. You cannot use the **Manual** and **Program Run** operating modes. You cannot activate host computer operation while running an NC program.

Secure connections for user

In the **Secure connections for user** area you activate the following functions:

Row	Meaning
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user.
Certificate management	In this row you open the **Certificate and keys** window. **Further information:** "SSH-secured DNC connection", Page 568

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	■ **Permit** The control permits network access without confirmation. ■ **Inquire** The control asks for confirmation upon a network access attempt. You can choose whether to permit or block the access once or always. ■ **Deny** The control does not permit any network access
Type	■ **Com1** Serial interface 1 ■ **Com2** Serial interface 2 ■ **Ethernet** Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

24.14 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: User's Manual for Programming and Testing

Requirement

- PostScript-capable printer
  The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting the PostScript emulation in the printer menu.
  Further information: "Note", Page 523

Description of function

The **Printer** menu item opens the **Heros Printer Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: User's Manual for Programming and Testing

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 523

There are various methods to start printing:

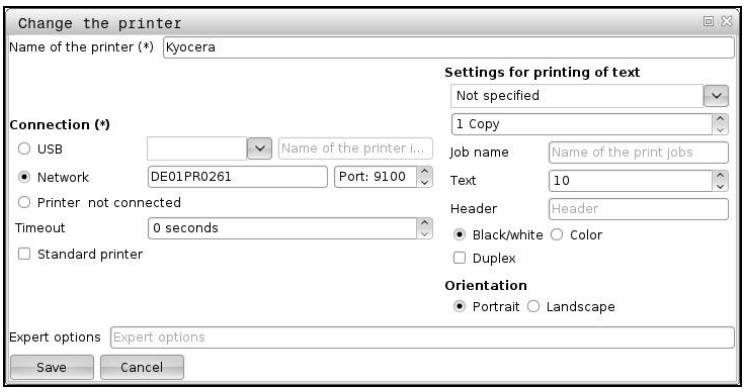
- Copying the file to be printed to the **PRINTER** drive
  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.
  You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.
- Using the **FN 16: F-PRINT** function

Buttons

The **Heros Printer Manager** window contains the following buttons:

Button	Meaning
Create	Creates a printer
CHANGE	Adapts the properties of the selected printer
COPY	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP	Selects a printer
DOWN	
STATUS	Displays the status information of the selected printer
PRINT A TEST PAGE	Prints a test page on the selected printer

Change the printer window



For each printer, the following properties can be set:

Setting	Meaning
Name of the printer	Customizes the printer name
Connection	Selects the connection ■ **USB**: The control automatically displays the name ■ **Network**: Network name or IP address of the printer Port for the network printer (default: 9001) ■ **Printer %1 not connected**
Timeout	Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**. Use this setting if the file to be printed is populated with FN functions (e.g., when probing).
Standard printer	Selects the default printer The control automatically assigns this setting to the first printer added.
Settings for printing of text	These settings are applicable when printing text documents: ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait or landscape for all printable files
Expert options	Available only to authorized specialists

24.14.1 Creating a printer

To create a new printer:

- Enter the printer name in the name dialog
- Select **Create**
- > The control creates a new printer.
- Press **CHANGE**
- > The control opens the **Change the printer** window.
- Define the properties
- Select **Save**
- > The control applies the settings and displays the defined printer in the list.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

24.15 VNC menu item

Application

VNC is software that shows the screen contents of a remote computer on a local computer, and also sends keyboard actions and mouse movements of the local computer to the remote computer.

Related topics

- Firewall settings

  Further information: "Firewall", Page 533
- Remote Desktop Manager (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 527

Description of function

The **VNC** menu item opens the **VNC settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Buttons and icons

The **VNC settings** window contains the following buttons and icons:

Button and icon	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set preferred owner of the focus	Activate the **Preferred owner of the focus** check box
	Another client owns the focus Mouse and keyboard are disabled
	You own the focus Entries can be made
	Prompt by another client to receive the focus Mouse and keyboard are disabled until the focus is assigned.

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.

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.

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.

VNC Focus Settings area

In the **VNC Focus Settings** area, you can make 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 in a focus change. Only if the **Enabling VNC focus** check box is active
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 active
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 active

Activate the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for purpose (e.g., ITC industrial computers).

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.
  Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	**VNC** is software with which another computer can be controlled over a network connection.

24.16 Remote Desktop Manager window (option 133)

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: "DNC menu item", Page 518

Requirement

- Remote Desktop Manager (software option 133)
- Existing network connection

  Further information: "Ethernet interface", Page 506

Description of function

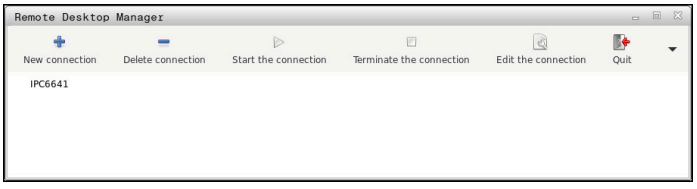
The **Remote Desktop Manager** menu item opens the **Remote Desktop Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

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 528
- **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control

  Further information: "VNC", Page 528
- **Switch-off/restart of a computer**: Automatically shut down a Windows computer together with the control
- **World Wide Web**: Only for authorized specialists
- **SSH**: Only for authorized specialists
- **XDMCP**: Only for authorized specialists
- **User-defined connection**: Only for 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	Use the **Edit the connection** window to create a new connection **Further information:** "Establishing and starting a connection", Page 531
Delete connection	Delete the selected connection
Start the connection	Start the selected connection **Further information:** "Establishing and starting a connection", Page 531
Terminate the connection	Terminate the selected connection
Edit the connection	Use the **Edit the connection** window to modify the selected connection **Further information:** "Connection settings", Page 529
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection **Further information:** "Exporting and importing connections", Page 532
Export the connections	Back-up the selected connection **Further information:** "Exporting and importing connections", Page 532

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 531

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

24.16.1 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

24.16.2 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 529
- Press **OK**

> The control saves the settings and closes the window.

- Select connection
- Select **Start the connection**

> The control starts the connection.

24.16.3 Exporting and importing connections

To export a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the desired connection
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Export the connections**
- > The control opens the **Select export file** window.
- ▶ Define the name of the saved file
- ▶ Select the target file
- ▶ Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Import connections**
- > The control opens the **Select file to import** window.
- ▶ Select file
- ▶ Select **Open**
- > The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!

If you do not shut down external computers properly, data may be irreversibly damaged or deleted.

- ▶ Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.

  With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

24.17 Firewall

Application

With the control you can set up a firewall for the primary network interface, and for a sandbox if needed. You can block incoming network traffic for specific senders and services.

Related topics

- Existing network connection
  Further information: "Ethernet interface", Page 506
- SELinux security software
  Further information: "SELinux security software", Page 502

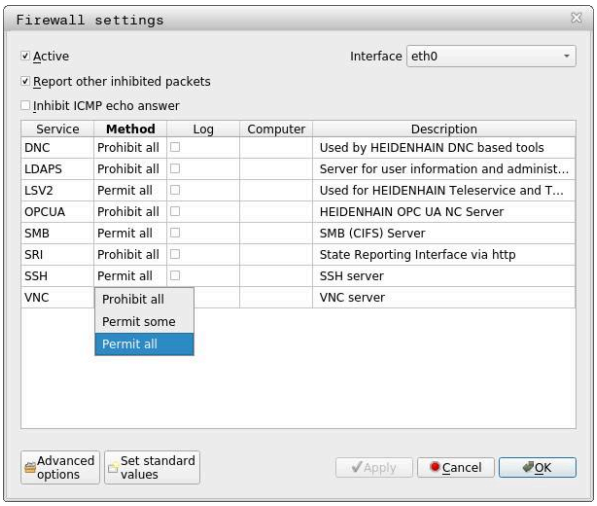
Description of function

The **Firewall** menu item opens the **Firewall settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

If you activate the firewall, the control displays an icon at the bottom right in the taskbar. The control displays the following icons, depending on the security level:

Icon	Meaning
	Firewall protection does not yet exist although it has been activated. Example: A dynamic IP address is used in the network interface configuration, but the DHCP server has not yet assigned an IP address. **Further information:** "DHCP server tab", Page 511
	Firewall active with medium security level.
	Firewall active with high security level. All services except for SSH are blocked.

Firewall settings



The **Firewall settings** window contains the following settings:

Setting	Meaning
Active	Activate or deactivate firewall
Interface	Select the interface ■ **eth0**: X26 of the control ■ **eth1**: X116 of the control ■ **brsb0**: Sandbox (optional) If a control has two Ethernet interfaces, then by default the DHCP server for the machine network is active for the second interface. With this setting you cannot activate the firewall for **eth1** because the firewall and DHCP server mutually exclude each other.
Report other inhibited packets	Activate the firewall with a high security level All services except for SSH are blocked.
Inhibit ICMP echo answer	If this check box is selected, the control does not respond to a ping request.
Service	Brief designation of services configured with the firewall. You can change the settings even if the services are not started. ■ **DNC** DNC server using the RPC protocol for external applications that were developed with RemoTools SDK (port 19003) For more detailed information, consult the RemoTools SDK manual. ■ **LDAPS** Server with user data and configuration of user administration ■ **LSV2** Functionality for **TNCremo**, TeleService, and other HEIDENHAIN PC tools (port 19000) ■ **OPC UA** Service provided by the **OPC UA NC Server** (port 4840). ■ **SMB** Only incoming SMB connections, meaning a Windows share on the control. Outgoing SMB connections are not influenced, meaning a Windows share connected to the control. ■ **SSH** SecureShell protocol (port 22) for secure LSV2 handling with active user administration; starting with HEROS 504 ■ **VNC** Access to screen contents. If you block this service, then not even TeleService programs from HEIDENHAIN can access the control. If you block this service, the control displays a warning in the **VNC settings** window. **Further information:** "VNC menu item", Page 523

Setting	Meaning
Method	Configure accessibility ■ **Prohibit all**: Cannot be accessed by anyone ■ **Permit all**: Can be accessed by everyone ■ **Permit some**: Can be accessed only by specific clients In the **Computer** column you must define the computer for which access is permitted. If you do not define a computer, the control activates **Prohibit all**.
Log	The control shows the following messages when transmitting network packets: ■ Red: Network packet blocked ■ Blue: Network packet accepted
Computer	IP address or host name of the computers with access rights. Separated by commas, if there are multiple computers The control converts the host name to an IP address when the control starts. If the IP address changes, you must restart the control or change the setting. The control issues an error message if it cannot convert the host name to an IP address. Only with the **Permit some** method
Advanced options	Only for network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

Notes

- Have your network specialist check and, if necessary, change the standard settings.
- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user identification.
- The firewall does not protect the second network interface **eth1**. Connect only trustworthy hardware to this interface, and do not use this interface for Internet connections.

24.18 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 533
- Network settings

  Further information: "Network configuration with Advanced Network Configuration", Page 586

Description of function

The **Portscan** menu item opens the **Portscan** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

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)

In the **Manual Execution** area, use the **Start** button to start the portscan manually. Use the **Automatic update on** function of the **Automatic Execution** to have the control automatically perform the portscan at a defined interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists my be open. The control shows a message window if a port is not listed.

24.19 Remote servicing

Application

In conjunction with the remote service setup tool, TeleService from HEIDENHAIN offers the possibility to establish encrypted end-to-end connections between a computer and a machine over the Internet.

Related topics

- External access
  Further information: "DNC menu item", Page 518
- Firewall
  Further information: "Firewall", Page 533

Requirements

- Existing Internet connection
  Further information: "Network configuration with Advanced Network Configuration", Page 586
- **LSV2** connection permitted by the firewall
  Remote diagnosis via the TeleService PC software uses the **LSV2** service. By default, the control's firewall blocks all incoming and outgoing connections. For this reason you must permit a connection for this service.
  You have the following options for permitting the connection:
  - Deactivate the firewall
  - Define the **Permit some** method for the **LSV2** service, and enter the name of the computer under **Computer**

  Further information: "Firewall", Page 533

Description of function

The **Remote Service** menu item opens the **HEIDENHAIN remote maintenance** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You need a valid session certificate for the servicing session.

Session certificate

During installation of the NC software, a temporary certificate is automatically installed on the control. A new installation or update may only be carried out by a service technician from the machine manufacturer.

A new certificate must be installed if no valid session certificate is installed on the control. Clarify with the service technician which certificate is needed. The service technician will then provide you with a valid certificate file, if necessary, which you must then install.

Further information: "Installing a session certificate", Page 538

In order to start the servicing session, you must enter the session key from the machine manufacturer.

24.19.1 Installing a session certificate

To install a session certificate on the control:

- Select the **Settings** application
- Select **Network/Remote Access**
- Double-tap or double-click **Network**
- > The control opens the **Network settings** window.
- Select the **Internet** tab

> The machine manufacturer defines the settings in the **Telemaintenance** field.

- Select **Add**
- > The control displays a selection menu.
- Select file
- Select **Open**
- > The control opens the certificate.
- Select **OK**
- Restart the control to load the settings

Notes

- If you've deactivated the firewall, you must reactivate it after the servicing session!
- If you permit the **LSV2** service in the firewall, the access security is ensured through the network settings. The network security is the responsibility of the machine manufacturer or the respective network administrator.

24.20 Backup and restore

Application

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the entire **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive
  Further information: User's Manual for Programming and Testing

Description of function

You open the backup function through the **NC/PLC Backup** menu item. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You open the restore function through the **NC/PLC Backup** menu item.

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 click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: User's Manual for Programming and Testing

Within the backup function you can chose between the following types of backups:

- **Back up the "TNC:" partition**
  Back-up all data on the **TNC:** drive
- **Back up the directory tree**
  Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**
  Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**
  Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** buttons.

24.20.1 Backing up data

To back-up the data of the **TNC:** drive:

- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the "TNC:" partition** window.
- ▶ Specify the type of backup
- ▶ Select **Forward**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select any predefined exclusion rules or ones you have defined yourself
- ▶ Select **Forward**
- > The control generates a list of files for backing up.
- ▶ Check list
- ▶ Deselect files if necessary
- ▶ Select **Forward**
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select **Forward**
- > The control generates the backup file.
- ▶ Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

24.20.2 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 **Forward**
> The control generates a list of files for restoring.
- Check list
- Deselect files if necessary
- Select **Forward**
- If necessary, pause the control with **Stop NC software**
- Select **Extract archive**
> The control restores the files.
- Confirm with **OK**
> The control restarts the NC software.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

24.21 Update documentation

Application

You can use the **Update documentation** function to install or update, for example, the **TNCguide** integrated product aid.

Related topics

- **TNCguide** integrated product aid
  Further information: "User's Manual as integrated product aid: TNCguide", Page 52
- Product aid on the HEIDENHAIN website
  TNCguide

Description of function

Settings ▶ **Diagnostics/Maintenance** ▶ **Update documentation**

In the **Update documentation** area, the control shows the file manager. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 541

The control shows all available documentation in the **Help** application.

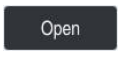
> In the **Update documentation** area you can install all HEIDENHAIN-specific documentation (e.g., NC error messages).

24.21.1 Transferring TNCguide

You can find and transfer the desired **TNCguide** version as follows:

- ▶ Select the link to the HEIDENHAIN website **TNCguide**
- ▶ Select **TNC Controls**
- ▶ Select **TNC7 Series**
- ▶ Select the NC software number
- ▶ Navigate to **Product help (HTML files)**
- ▶ Select **TNCguide** in the desired language
- ▶ Select path to save the file
- ▶ Select **store**
- > The download begins.
- ▶ Transfer the downloaded file to the TNC control

- ▶ Select the **Home** operating mode
- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Select **Update documentation**
- > The control opens the **Select installation file** area.
- ▶ Select the desired file with extension ***.tncdoc**



- ▶ Select **Open**
- > A pop-up window appears, stating whether the installation was successful or failed.
- ▶ Select the **Help** application

- ▶ Select **home**
- > The control shows all available documentation.

24.22 TNCdiag

Application

The control displays status and diagnostic information of HEIDENHAIN components in the **TNCdiag** window.

Description of function

	Only use this function after consultation with your machine manufacturer.

	For more information, please refer to the **TNCdiag** documentation.

24.23 Machine parameters

Application

You can configure the behavior of the control with machine parameters. The control offers the applications **MPs for users** and **MPs for setters** for this. You can open the **MPs for users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

- List of machine parameters in the **MPs for setters** application
  Further information: "Machine parameters", Page 592

Requirements

- Code number 123
  Further information: "Code numbers", Page 493
- Contents of the **MPs for setters** application are defined by the machine manufacturer

Description of function

The **MPs for setters** menu item opens the **MPs for setters** application. The menu item is in the **Machine parameters** group of the **Settings** application.

In the **Machine parameters** group the control shows only those menu items that you can choose with the current access rights.

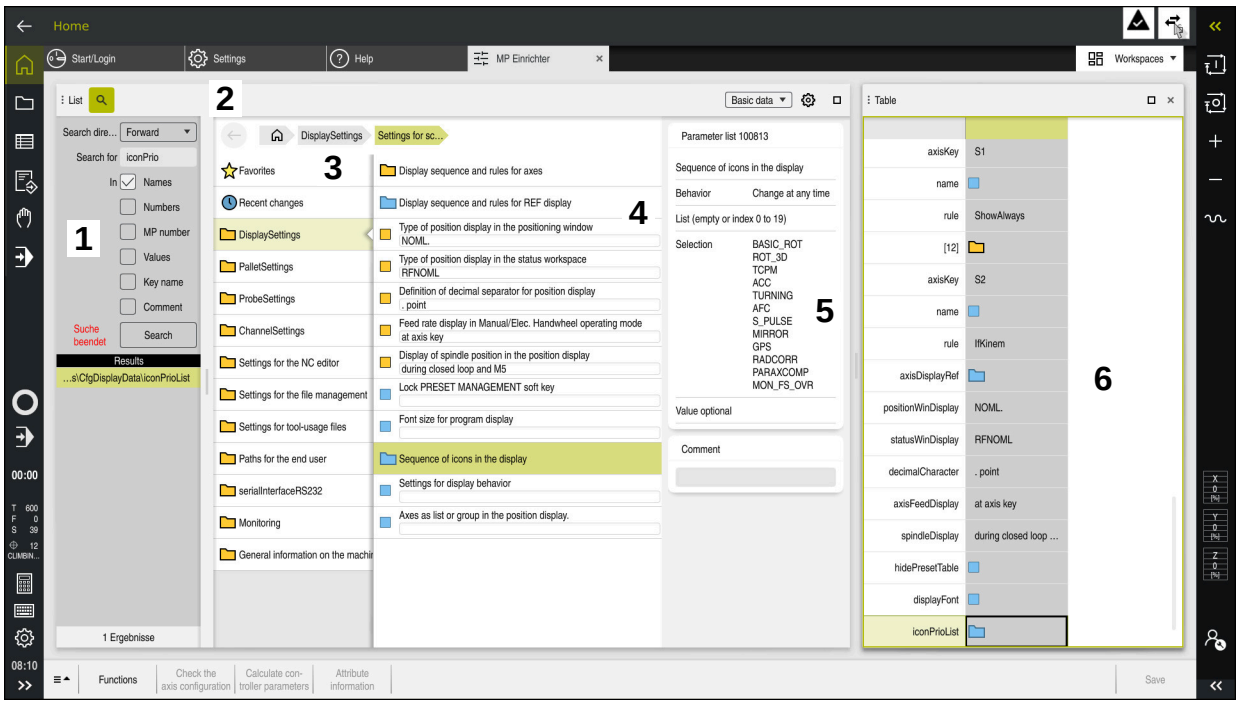
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **List**
- **Table**

You can't close the **List** workspace.

Areas of the configuration editor



MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 **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

You can show and hide the **Search** column, use a selection menu to filter the contents, and open the **Configuration** window.

Further information: "Configuration window", Page 546

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 Information area

The control displays information about the selected machine parameter or change.

Further information: "Information area", Page 546

6 **Table** workspace

In the **Table** workspace the control shows the selected contents within the structure. In order to do so, in the **Configuration** window the **Synchronized navigation in list and table** toggle switch must be set to active.

The control displays the following information:

- Name of the objects
- Icon of the objects
- Value of the machine parameters

Icons and buttons

The configuration editor contains the following icons and buttons:

Icon or button	Meaning
	Open the **Configuration** window **Further information:** "Configuration window", Page 546
	Select **Recent changes**
	Object exists ■ Data object ■ Directory ■ Parameter list
	Object empty
	Machine parameter exists
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter unreadable and uneditable
	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** User's Manual for Programming and Testing
Check the axis configuration	Only for the machine manufacturer
Calculate controller parameters	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

Configuration window

In the **Configuration** window you define the settings for displaying the machine parameters in the configuration editor.

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

- **List**
- **Table**

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

Setting	Meaning
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.
Show details	Hide or show the information area with this toggle switch.

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

Setting	Meaning
Show details when table displayed	If the toggle switch is active, the control shows the information area even if the **Table** workspace is open. If the toggle switch is not active, the control shows the information area only if the **Table** workspace is closed.
Synchronized navigation in list and table	If the toggle switch is active, the control always shows in the **Table** workspace the object that is marked in the **List** workspace, and vice versa. If the toggle switch is not active, the contents of the two workspaces do not synchronize.

Information area

If you select contents from the favorites or the structure, the control will display some of the information below in the information area:

- Type of object, such as data object list or parameter, perhaps with number
- Descriptive text of machine parameter
- Information about the effect
- Permitted or required input
- Behavior, such as program run disabled
- MP number of the iTNC 530 for machine parameters
- Machine parameter optional

If you select content from one of the recent changes, the control will display the information below in the information area:

- Sequential number of the change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Information about the effect

24.24 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is individually adapted.

Related topics

- Workspaces
  Further information: "Workspaces", Page 81
- Control interface
  Further information: "Areas of the control's user interface", Page 78

Description of function

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

You manage the configurations in the **Settings** application.

To navigate to this function:

Settings ▶ **Configurations** ▶ **Configurations**

The **Configurations** area contains the following functions:

Function	Meaning
Active configuration	Activate a configuration from a selection menu **Further information:** "Desktop menu workspace", Page 93
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 uses the **Save** button to overwrite the **OEM configuration**.

The control displays the following information about all available configurations in a table:

Column	Meaning
Configuration name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the **Active configuration** selection menu.
Exportable	If this toggle switch is active, you can export the configuration. **Further information:** "Exporting and importing configurations", Page 548
Edit	This column contains two buttons, for renaming and deleting the configuration.

Press the **Add** button to create a new configuration.

24.24.1 Exporting and importing configurations

To export configurations:

- Select the **Settings** application
- Select **Configurations**

> The control opens the **Configurations** area.

- Activate the **Exportable** toggle switch for the desired configuration, if necessary

Export

- Select **Export**

> The control opens the **Save as** window.

- Select the target file
- Enter a file name

Create

- Select **Create**

> The control saves the configuration file.

To import configurations:

Import

- Select **Import**

> The control opens the **Import configurations** window.

- Select file

Import configuration

- Select **Import configuration**

> 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**.

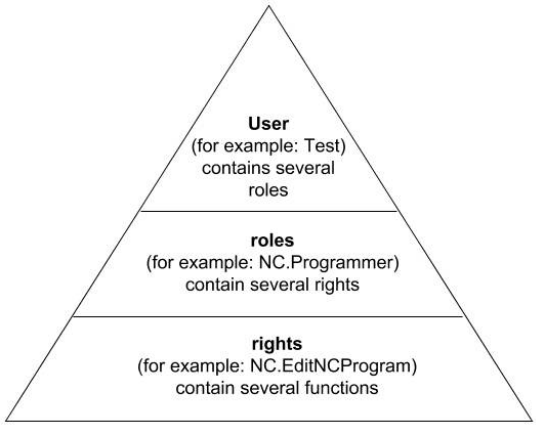
Description of function

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 550
- Roles
  Further information: "Roles", Page 552
- Rights
  Further information: "Rights", Page 552



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 556

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 557

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **useradmin**

  The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.

- **sys**

  The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.

- **user**

  In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.

- **oem**

  The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer defines function users who are required for specific tasks such as machine maintenance.

By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.

Further information: "Active user window", Page 558

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 603

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 568

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 607

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 560

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 any logged-on user.

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 Opening the **User management** 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 management** window after each configuration step.
- If you exit the **User management** window directly after having activated user administration, the control will prompt you for a restart once.

Opening the User management window

To open the **User management** window:

- ▶ Select the **Settings** application
- ▶ Select **Operating system**
- ▶ Double-tap or double-click **CurrentUser**
- > The control opens the **User management** window in the **Settings** tab.

  Further information: "User management window", Page 558

Activating user administration

To activate user administration:

- ▶ Select **User administration active**
- > The control shows the message **Password for user 'useradmin' missing**.
- ▶ Retain or reactivate the active status of the **Anonymize users in login data** function

- The purpose of the **Anonymize users in login 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 management** window directly after having activated user administration, the control will prompt you for a restart once.

Define the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 550

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 553

- Repeat the password
- Select **Set new password**
- > The control shows the message **Settings and password for 'useradmin' were changed**.

Setting up a database

To set up a database:

- Select a database for saving the user data, such as **Local LDAP database**
- Select **Configuration**
- > The control opens a window for configuring the corresponding database.
- Follow the instructions from the control in the window
- Select **APPLY**

> The following options are available for saving your user data:
> - **Local LDAP database**
> - **LDAP on remote computer**
> - **Connection to Windows domain**
>
> Parallel operation of Windows users and users from an LDAP database is possible.
> **Further information:** "Saving user data", Page 560

Creating a new user

To create a new user:

- Select the **User management** tab
- Select **Create new user**
> The control adds a new user to the **User list**.
- Change the name as needed
- Edit a password as needed
- Define a profile image as needed
- Enter a description as needed
- Select **Add role**
> The control opens the **Add role** window.
- Select a role
- Select **Add**

> You can also add roles using the **Add external login** and **Add local login** buttons.
>
> **Further information:** "Roles", Page 552

- Select **Close**
> The control closes the **Add role** window.
- Select **OK**
- Select **APPLY**
> The control adopts the changes.
- Select **END**
> The control opens the **System reboot required** window.
- Select **Yes**
> The control restarts.

> The user must change the password when logging in for the first time.

25.1.2 Deactivating user administration

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 550

To deactivate user administration:

- ▶ Log in as a function user
- ▶ Opening the **User management** window
- ▶ Select **User administration inactive**
- ▶ If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
- ▶ Select **APPLY**
- ▶ Select **END**
- > The control opens the **System reboot required** window.
- ▶ Select **Yes**
- > The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in login 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 login 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 574
- 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: "Remote Desktop Manager window (option 133)", Page 527

25.2 User management window

Application

In the **User management** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- **Active user** window
  Further information: "Active user window", Page 558

Requirement

- If user administration is active, the HEROS.Admin role
  Further information: "List of roles", Page 603

Description of function

To navigate to this function:

Settings ▶ **Operating system** ▶ **UserAdmin**

The **User management** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration **Further information:** "Configuring user administration", Page 554
User management	Create or remove users, change rights, add profile images **Further information:** "Creating a new user", Page 556
Password settings	Define password requirements **Further information:** "Password settings", Page 553
User-defined roles	Roles created for a Windows domain **Further information:** "Connection to Windows domain", Page 562

25.3 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 568
- Logon with smartcards
  Further information: "Logon with smartcards", Page 566
- Available roles and rights
  Further information: "User administration roles and rights", Page 603

Description of function

To navigate to this function:

Settings ▶ Operating system ▶ Current User

By default the **Active user** window is on the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the **Added rights** tab, enable rights from another user or function user until the next logoff
Open user administration	Open the **User management** window **Further information:** "User management window", Page 558
SSH keys and certificates	Manage keys and certificates for client connections **Further information:** "SSH-secured DNC connection", Page 568 **Further information:** "OPC UA NC Server (options 56 to 61)", Page 513
Create token **Delete token**	Manage smartcards for logon with a card reader **Further information:** "Logon with smartcards", Page 566
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 553

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 550

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 560
- **LDAP on remote computer**
  Further information: "LDAP database on a remote computer", Page 561
- **Connection to Windows domain**
  Further information: "Connection to Windows domain", Page 562

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 561
- Connecting a Windows domain with user administration
  Further information: "Connection to Windows domain", Page 562

Requirements

- User administration is active
  Further information: "Activating user administration", Page 554
- **useradmin** user is logged on
  Further information: "Users", Page 550

Description of function

A local LDAP database offers the following options:

- Using user administration on one single control
- Setting up a central LDAP server for more than one control
- Exporting an LDAP server configuration file if the exported database is to be used by more than one control

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- Opening the **User management** 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 560
- Connecting a Windows domain with user administration

  Further information: "Connection to Windows domain", Page 562

Requirements

- User administration is active

  Further information: "Activating user administration", Page 554
- **useradmin** user is logged on

  Further information: "Users", Page 550
- 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 562

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:

- ▶ Opening the **User management** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Export server configuration**
- > The control opens the **Export LDAP configuration file window.**
- ▶ Enter the name for the server configuration file into the name field
- ▶ Save the file to the desired folder
- > The control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:

- ▶ Opening the **User management** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **LDAP on remote computer**
- ▶ Select **Import server configuration**
- > The control opens the **Import LDAP configuration file window.**
- ▶ Select the existing configuration file
- ▶ Select **FILE**
- ▶ Select **APPLY**
- > The control imports the configuration file.

25.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function you can connect the data of a domain controller with the control's user administration.

Related topics

- Configuring an LDAP database on a control
  Further information: "Local LDAP database", Page 560
- Using an LDAP database on multiple controls
  Further information: "LDAP database on a remote computer", Page 561

Requirements

- User administration is active
  Further information: "Activating user administration", Page 554
- **useradmin** user is logged on
  Further information: "Users", Page 550
- Windows domain controller present in the network
- You have access to the password of the domain controller
- You have access to the user interface of the domain controller, perhaps supported by an IT administrator
- Domain controller accessible in the network

Description of function

Use the **Configuration** function to configure the connection:

- Use the **Map SIDs to Unix UIDs** check box to select whether Windows SIDs are automatically mapped to Unix UIDs
- Use the **Use LDAPs** check box to select LDAP or secure LDAPs. For LDAPs, define whether or not the secure connection verifies a certificate
- Define a special group of Windows users to whom you want to restrict the connection to this control
- Modify the organizational unit in which the HEROS role names are stored
- 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)
- Modify the separator within the HEROS role names

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** function to export the roles to an ***.ldif** file

There are the following ways to create groups corresponding to the different roles:

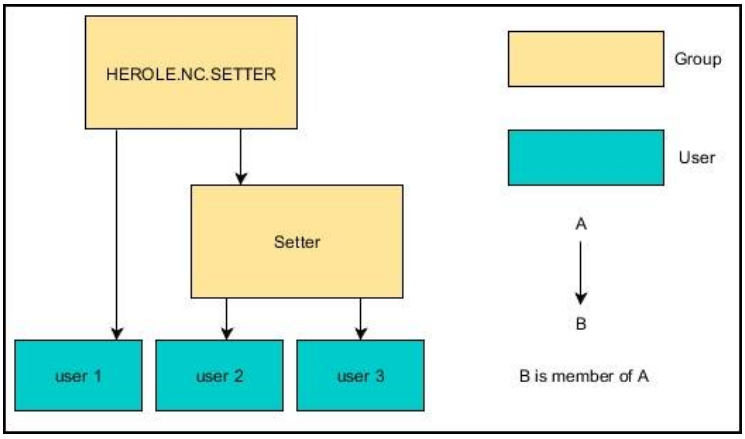
- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

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.

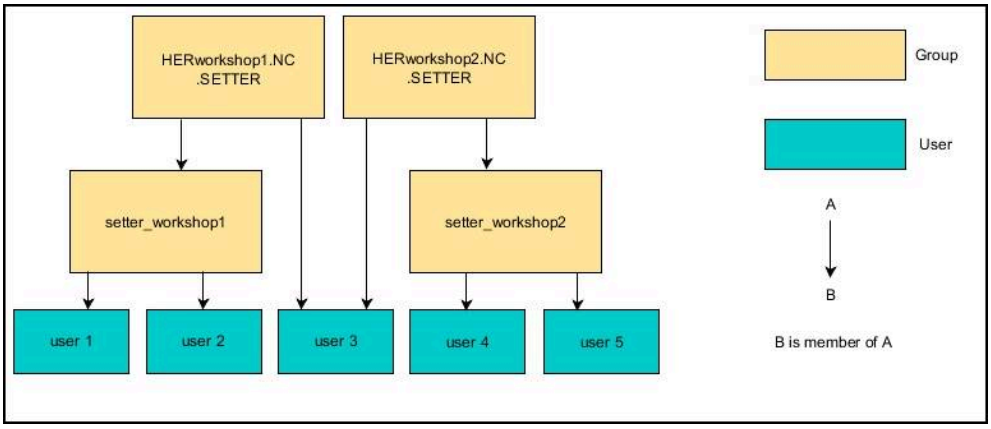
Example 1

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:



Setting up the Connection to Windows domain function

To set up a **Connection to Windows domain**:

- Opening the **User management** window
- Select **Connection to Windows domain**
- Select **Find domain**
- > The control selects a domain.
- Select **APPLY**
- > The control opens the **Connect to domain** window.

> With the **Organizational unit for computer account:** function, you can specify in which of the already existing organizational units you want to create the access, e.g.
>
> - ou=controls
> - cn=computers
>
> The values you enter must match the conditions of the domain. The terms are not exchangeable.

- Enter the user name of the domain controller
- Enter the password of the domain controller
- Confirm your input
- > The control connects to the Windows domain found.
- > The control checks whether all of the required roles have been created in the domain as groups.
- Add groups, if necessary

  Further information: "Groups of the domain", Page 563

25.5 Autologin in user administration

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

- User login
  Further information: "Logging on with user administration", Page 565
- Configure user administration
  Further information: "Configuring user administration", Page 554

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 management** window you can define a user for autologin.

Further information: "User management window", Page 558

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 567

25.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

- Automatic user logon
  Further information: "Autologin in user administration", Page 565

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 567

Description of function

The control displays the Login dialog in the following cases:

- After the **User logout** function has been executed
- After the **Switch user** function has been executed
- After the screen has been locked by the **screensaver**
- Immediately after control startup if user administration is active and **Autologin** is not enabled

Further information: "HEROS menu", Page 574

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 567

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

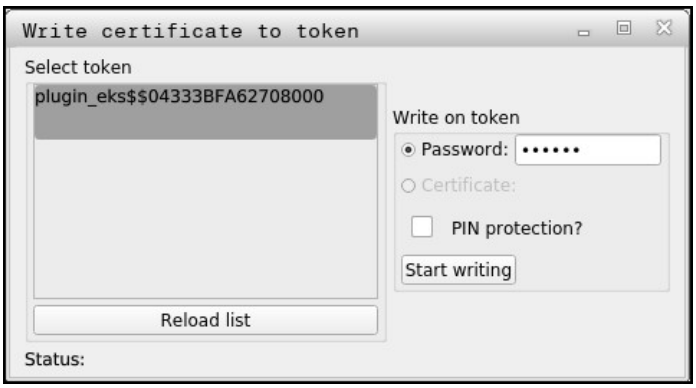
> The control shows in the Login dialog whether CAPS LOCK is active.

- > 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 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.

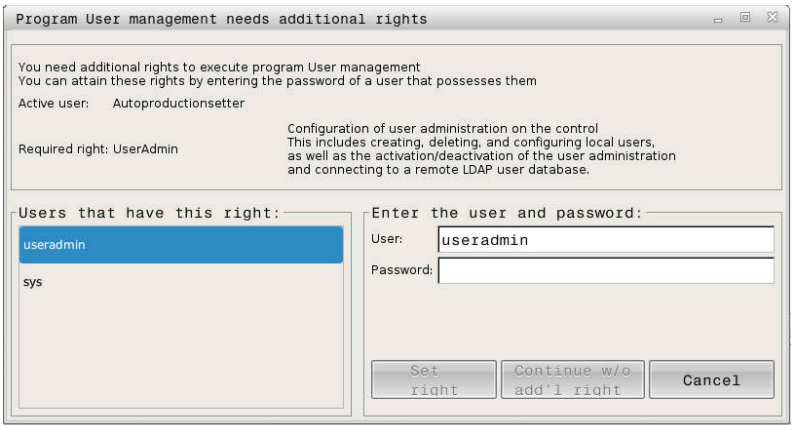
Related topics

- Temporarily granting additional rights in the **Active user** window
  Further information: "Active user window", Page 558

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 533
- Roles for remote logon
  Further information: "Roles", Page 552

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

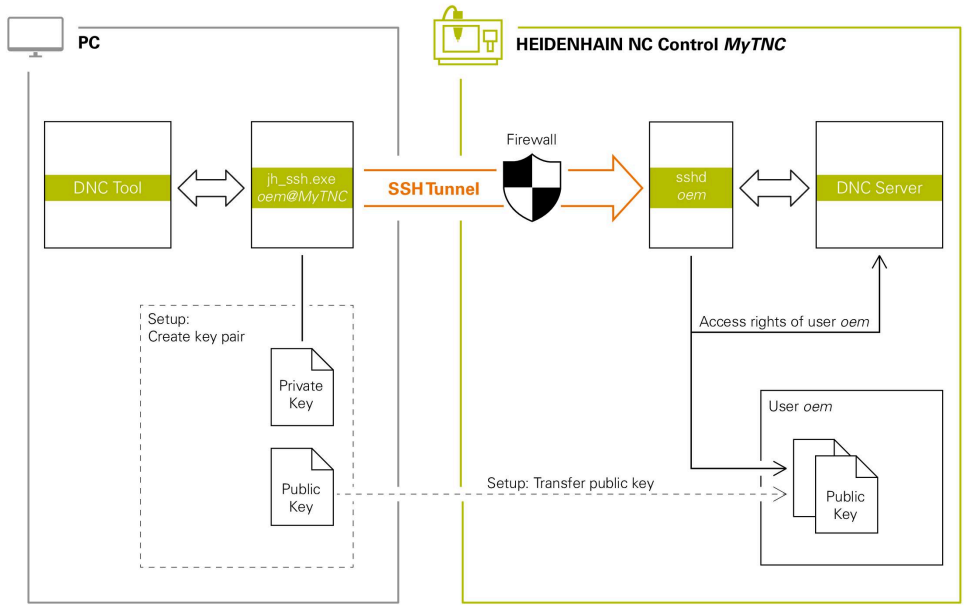
Description of function

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 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 (options 56 to 61)", Page 513
- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user identification.

  The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transmit a public key to the control using a USB device or a 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 needing to transmit 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

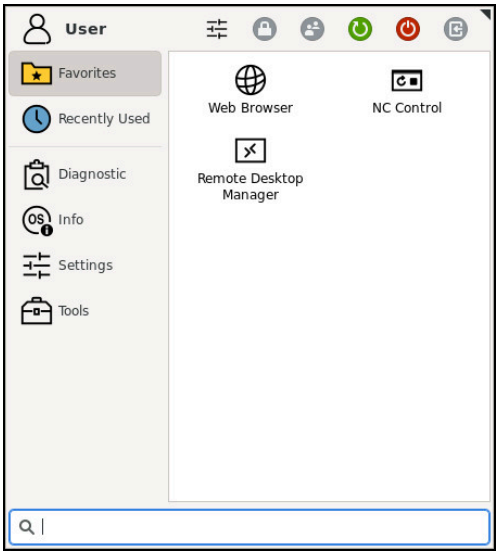
Related topics

- Opening HEROS functions through the **Settings** application
  Further information: "Settings Application", Page 489

Description of function

You open the HEROS menu with the green DIADUR icon in the taskbar or with the **DIADUR** key.

Further information: "Taskbar", Page 578



Standard view of the HEROS menu

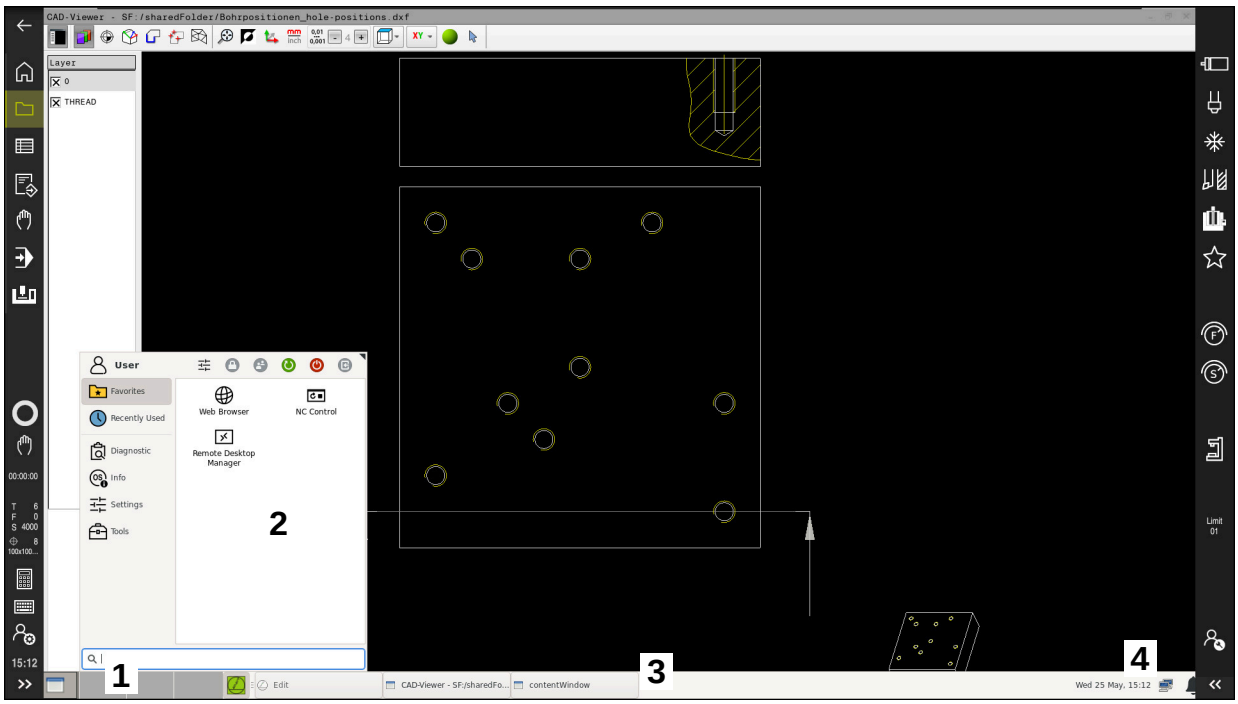
The HEROS menu contains the following functions:

Area	Function
Header	■ User name **Further information:** "Active user window", Page 558 ■ 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 549
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HeLogging**: Define settings for internal diagnostic files ■ **HeMenu**: Available only to authorized specialists ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 536 ■ **Portscan OEM**: Available only to authorized specialists ■ **RemoteService**: Start and stop remote maintenance **Further information:** "Remote servicing", Page 537 ■ **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 542 ■ **TNCscope** Software for data recording

Area	Function
Settings	■ **Screensaver**: Screensaver ■ **Current User** **Further information:** "Active user window", Page 558 ■ **Date/Time** **Further information:** "Adjust system time window", Page 500 ■ **Firewall** **Further information:** "Firewall", Page 533 ■ **HePacketManager**: Available only to authorized specialists ■ **HePacketManager Custom**: Available only to authorized specialists ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 501 ■ **Network** **Further information:** "Ethernet interface", Page 506 ■ **OEM Function Users** **Further information:** "User administration", Page 549 ■ **OPC UA NC Server Connection Assistant** **Further information:** "OPC UA connection wizard function (options 56 to 61)", Page 516 ■ **OPC UA NC Server License** **Further information:** "OPC UA license settings function (options 56 to 61)", Page 517 ■ **PKI Admin**: Manage certificates for the control (e.g., for **OPC UA NC Server**) "OPC UA NC Server (options 56 to 61)" ■ **Printer** **Further information:** "Printers", Page 520 ■ **SELinux** **Further information:** "SELinux security software", Page 502 ■ **Shares** **Further information:** "Network drives on the control", Page 503 ■ **UserAdmin** **Further information:** "User management window", Page 558 ■ **VNC** **Further information:** "VNC menu item", Page 523 ■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 579
Info	■ **About HeROS**: Open information about the operating system of the control ■ **About Xfce**: Open information on the Window manager

Area	Function
Tools	■ **Switch-off**: Shut-down or restart ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Document Viewer**: Display and print files (e.g., PDF files) ■ **Geeqie**: Open, manage, and print graphics ■ **Gnumeric**: Open, edit, and print tables ■ **IDS Camera Manager**: Manage cameras connected to the control ■ **keypad horizontal**: Open virtual keyboard ■ **keypad vertical**: Open virtual keyboard ■ **Leafpad**: Open and edit text files ■ **NC Control**: Start or stop the NC software independently of the operating system ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 538 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 538 ■ **QupZilla**: Alternative web browser for touch operation ■ **Real VNC Viewer**: Define the settings for external software accessing the control (e.g., for maintenance purposes) ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 527 ■ **Ristretto**: Open graphics ■ **TNCguide**: Open help files in CHM format ■ **TouchKeyboard**: Open keyboard for touch operation ■ **Web Browser**: Start the web browser ■ **Xarchiver**: Extract or compress directories
Searching	Full-text search 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 574

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 533
- Network status

  Further information: "Ethernet interface", Page 506
- 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 the **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 78

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 78
- 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 542

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: **BAUD_110**, **BAUD_150**, **BAUD_300 BAUD_600**, **BAUD_1200**, **BAUD_2400**, **BAUD_4800**, **BAUD_9600**, **BAUD_19200**, **BAUD_38400**, **BAUD_57600**, **BAUD_115200**
protocol (no. 106702)	Communications protocol ■ **STANDARD**: Standard data transmission, line-by-line ■ **BLOCKWISE**: Packet-based data transfer ■ **RAW_DATA**: Transmission without protocol (purely character-by-character) Input: **STANDARD**, **BLOCKWISE**, **RAW_DATA**
dataBits (no. 106703)	Data bits in each transferred character Input: **7 Bit**, **8 Bit**
parity (no. 106704)	Parity bit used to check for transmission errors ■ **NONE**: No parity, no error detection ■ **EVEN**: Even parity, error if the number of bits set is odd ■ **ODD**: Odd parity, error if the number of bits set is even Input: **NONE**, **EVEN**, **ODD**
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission. Input: **1 Stop-Bit**, **2 Stop-Bits**
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. ■ **NONE**: No data-flow check ■ **RTS_CTS**: Hardware handshaking, transmission stop is active through RTS ■ **XON_XOFF**: Software handshaking, transmission stop is active through DC3 Input: **NONE**, **RTS_CTS**, **XON_XOFF**
fileSystem (no. 106707)	File system for the serial interface ■ **EXT**: Minimum file system for printers or non-HEIDENHAIN transmission software ■ **FE1**: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: **EXT**, **FE1**
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ **TRUE**: The BCC does not correspond to any control character ■ **FALSE**: Function not active Input: **TRUE**, **FALSE**

Machine parameters	Setting
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):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Type of handshake	RTS_CTS
File system for file operations	FE1

TNCserver is part of the TNCremo software for PCs.

Further information: "PC software for data transfer", Page 581

26.4 PC software for data transfer

Application

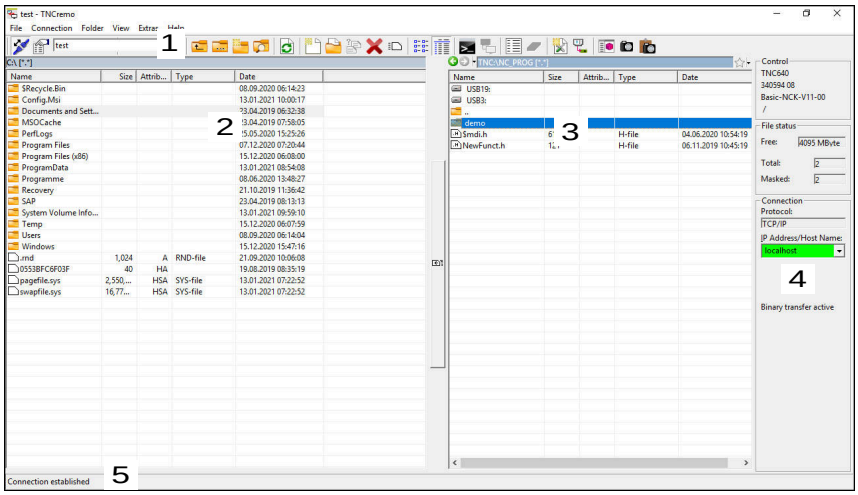
HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Requirements

- PC operating system:
  - Windows 7
  - Windows 8
  - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- An available serial interface or connection to the control's network

Description of function

The TNCremo data transfer software provides the following areas:



1 Toolbar

This area provides the most important TNCremo functions.

2 File list of PC

In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).

3 File list of control

In this area, TNCremo displays all of the folders and files of the connected drive of the control.

4 Status display

In the status display, TNCremo shows information about the current connection.

5 Connection status

The connection status indicates whether a connection is currently active.

For more information, refer to the integrated help system of TNCremo.
You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user identification. The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

  The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).
- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

26.5 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: User's Manual for Programming and Testing

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 538

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 581
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 503
- External data carriers
  You can back-up files to external data carriers or use external data carriers to transfer the files.
  Further information: User's Manual for Programming and Testing

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.

26.6 Opening files with additional software

Application

The control provides several additional software programs for opening and editing standard file types:

Related topics

- File types
  Further information: User's Manual for Programming and Testing

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS) CSV	Gnumeric
INI A TXT	Leafpad
HTM/HTML	Web browser For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto or Geeqie Ristretto can only open graphics files. Geeqie can also edit and print graphics.
OGG	Parole With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file with the correct tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

26.6.1 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 502

26.7 Network configuration with Advanced Network Configuration

Application

Using the **Advanced Network Configuration**, you can add, edit or remove profiles for the network connection.

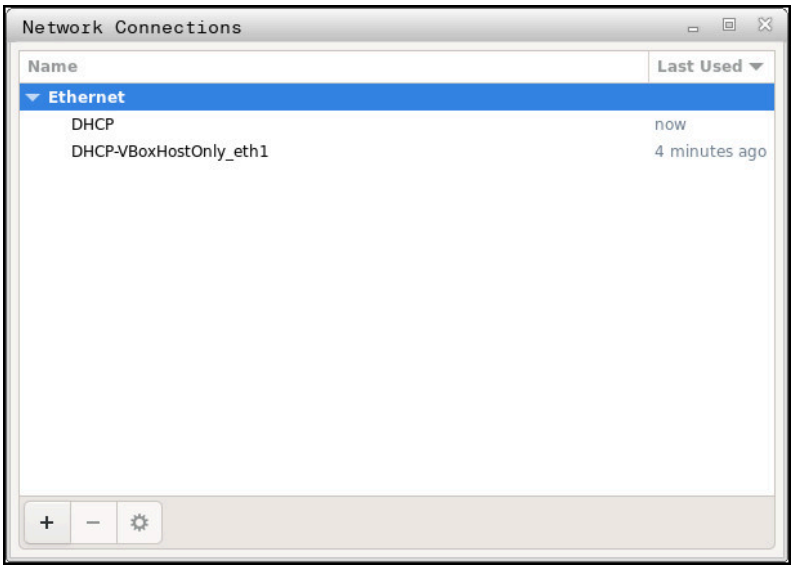
Related topics

- Network settings

  Further information: "Editing network connection window", Page 587

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.



Network Connections window

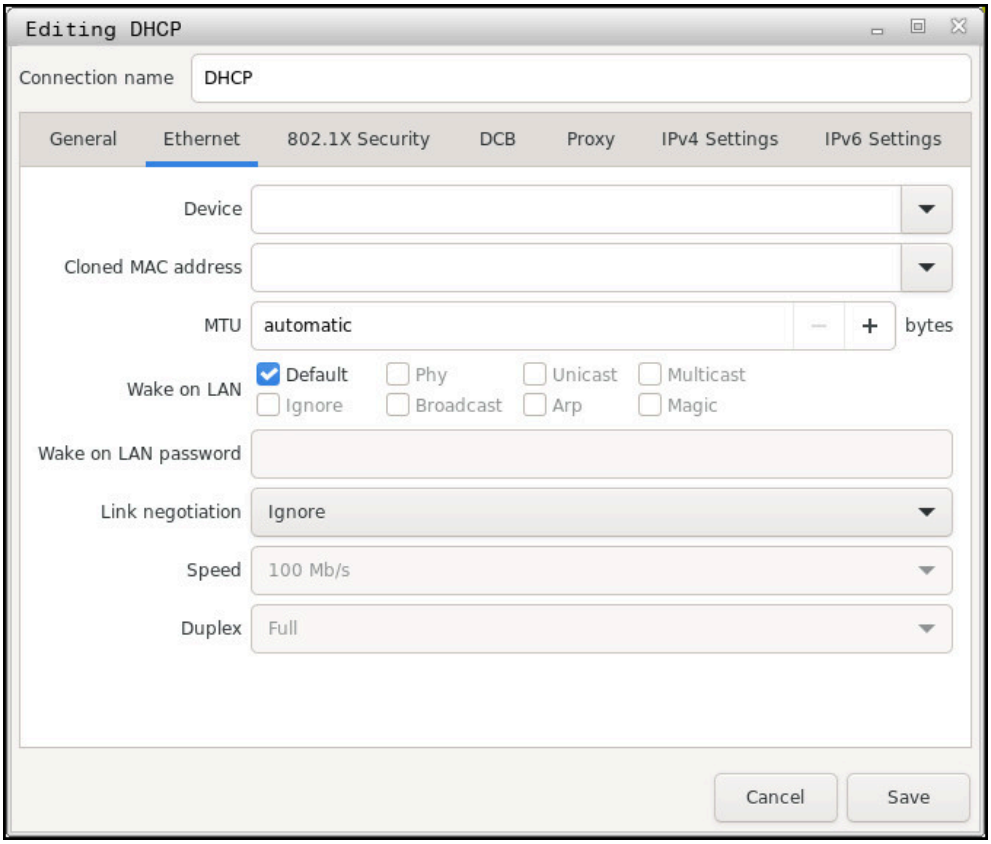
Symbols in the Network connections window

The following symbols are shown in the **Network connections** window:

Symbol	Function
+	Add network connection
−	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "Editing network connection window", Page 587

26.7.1 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.



Editing network connection window

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

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
	Currently no function
	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 **Manual** is selected Selection by means of a selection window
Full duple_x	Here you have to select the duplex setting: ■ ■ Only if **Manual** is selected Selection by means of a selection window

802.1X Security tab

Currently no function

DCB tab

Currently no function

Proxy tab

Currently no function

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 ser_vers:	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**
Addition-al s_earch domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **_Method: Manual**
D_HCP client ID:	Currently no function
Require IPv_4 addressing for this connection to complete	Currently no function

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 Machine parameters

The following list shows the machine parameters that you can edit with the code number 123.

Related topics

- Editing machine parameters with the **MPs for setters** application
  Further information: "Machine parameters", Page 542

27.2.1 List of user parameters

Refer to your machine manual.

- The machine manufacturer can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.
- The machine manufacturer can adapt the structure and contents of the user parameters. The display on your machine may be different.

Depiction in the configuration editor	MP number
DisplaySettings	
CfgDisplayData Settings for screen displays	100800
axisDisplay Display sequence and display rules for axes	100810
x	
axisKey Key name of the axis	100810. [Index].01501
name Axis designation	100810. [Index].01502
rule Display rule for the axis	100810. [Index].01503
axisDisplayRef Sequence and rules for display axes before crossing the reference marks	100811
x	
axisKey Key name of the axis	100811. [Index].01501
name Axis designation	100811. [Index].01502
rule Display rule for the axis	100811. [Index].01503
positionWinDisplay Type of position display in the position window	100803
statusWinDisplay Type of position display in the Status workspace	100804
decimalCharacter Definition of the decimal separator for the position display	100805
axisFeedDisplay Display of the feed rate in the applications of the Manual operating mode	100806
spindleDisplay Display of spindle position in the position display	100807

Depiction in the configuration editor	MP number
hidePresetTable Disable the PRESET MANAGEMENT soft key	100808
displayFont Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.	100812
iconPrioList Sequence of icons in the display	100813
compatibilityBits Settings for display behavior	100815
axesGridDisplay Axes as list or group in the position display.	100806
CfgPosDisplayPace Display step for the individual axes	101000
xx	
displayPace Display step for position display in [mm] or [°]	101001
displayPaceInch Display step for position display in [inch]	101002
CfgUnitOfMeasure Definition of unit of measure in effect for display	101100
unitOfMeasure Unit of measure for display and user interface	101101
CfgProgramMode Format of the NC programs and cycle display	101200
programInputMode MDI: Program entry in HEIDENHAIN Klartext format or ISO format	101201
CfgDisplayLanguage Definition of the NC and PLC conversational language	101300
ncLanguage NC conversational language	101301
applyCfgLanguage Load the language of the NC control	101305
plcDialogLanguage PLC conversational language	101302
plcErrorLanguage PLC error message language	101303
helpLanguage Language for online help	101304
CfgStartupData Behavior during control startup	101500

Depiction in the configuration editor	MP number
powerInterruptMsg Acknowledge the Power interrupted message	101501
opMode Operating mode that is switched to when the control has fully booted	101503
subOpMode Submode to be activated for the operating mode entered in 'opMode'	101504
CfgClockView Display mode for time of day	120600
displayMode Display mode for time of day on the screen	120601
timeFormat Time format of digital clock	120602
CfgInfoLine Link row on/off	120700
infoLineEnabled Enable/disable info line	120701
CfgGraphics Settings for 3D simulation graphics	124200
modelType Model type of the 3D simulation graphics	124201
modelQuality Model quality of the 3D simulation graphics	124202
clearPathAtBlk Reset tool paths for new BLK FORM	124203
extendedDiagnosis Write graphics journal files after restart	124204
CfgPositionDisplay Settings for the digital readout	124500
progToolCallDL Position display with TOOL CALL DL	124501
CfgTableEditor Table editor configuration	125300
deleteLoadedTool Behavior when deleting tools from the pocket table	125301
indexToolDelete Behavior when deleting a tool's index entries	125302
showResetColumnT Show the RESET COLUMN T soft key	125303
CfgDisplayCoordSys Setting the coordinate systems for the display	127500

Depiction in the configuration editor	MP number
transDatumCoordSys Coordinate system for the datum shift	127501
CfgGlobalSettings GPS display settings	128700
enableOffset Show offset in the GPS dialog	128702
enableBasicRot Show an additive basic rotation in the GPS dialog	128703
enableShiftWCS Show shift of W-CS in the GPS dialog	128704
enableMirror Show mirroring in the GPS dialog	128712
enableShiftMWCS Show shift of mW-CS in the GPS dialog	128711
enableRotation Show rotation in the GPS dialog	128707
enableFeed Show feed rate in the GPS dialog	128708
enableHwMCS M-CS coordinate system is selectable	128709
enableHwWCS W-CS coordinate system is selectable	128710
enableHwMWCS mW-CS coordinate system is selectable	128711
enableHwWPLCS WPL-CS coordinate system is selectable	128712
enableHwAxisU U axis can be selected	128709
enableHwAxisV V axis can be selected	128709
enableHwAxisW W axis can be selected	128709
CfgRemoteDesktop Settings for Remote Desktop connections	100800
connections List of Remote Desktop connections to be displayed	133501
autoConnect Start connection automatically	133505
title Name of the OEM operating mode	133502
dialogRes Name of a text	133502.00501

Depiction in the configuration editor	MP number
text Language-sensitive text	133502.00502
icon Path/name for optional icon graphic file	133503
locations List with positions where this Remote Desktop connection is displayed	133504
x	
opMode Operating mode	133504. [Index].133401
subOpMode Optional submode for the operating mode specified in 'opMode'	133504. [Index].133402
PalletSettings	
CfgPalletBehaviour Behavior of the pallet control cycle	202100
failedCheckReact Specify reaction to program check and tool check	202106
failedCheckImpact Specify effect of program check or tool check	202107
ProbeSettings	
CfgTT Configuration of the tool calibration	122700
TT140_x	
spindleOrientMode M function for spindle orientation	122704
probingRoutine Probing routine	122705
probingDirRadial Probing direction for tool radius measurement	122706
offsetToolAxis Distance from lower edge of tool to upper edge of stylus	122707
rapidFeed Rapid traverse in probing cycle for TT tool touch probe	122708
probingFeed Probing feed rate for tool measurement with non-rotating tool	122709
probingFeedCalc Calculation of the probing feed rate	122710

Depiction in the configuration editor	MP number
spindleSpeedCalc Speed determination method	122711
maxPeriphSpeedMeas Maximum permissible surface speed of the tool edge for radius measurement	122712
maxSpeed Maximum permissible speed during tool measurement	122714
measureTolerance1 Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)	122715
measureTolerance2 Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)	122716
stopOnCheck NC stop during tool check	122717
stopOnMeasurement NC stop during tool measurement	122718
adaptToolTable Change the tool table during tool check and tool measurement	122719
CfgTTRoundStylus Configuration of a round stylus	114200
TT140_x	
centerPos Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum	114201
safetyDistToolAx Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction	114203
safetyDistStylus Safety zone around the stylus for pre-positioning	114204
CfgTTRectStylus Configuration of a rectangular stylus	114300
TT140_x	
centerPos Coordinates of the stylus center	114313
safetyDistToolAx Set-up clearance above the stylus for pre-positioning	114317

Depiction in the configuration editor	MP number
safetyDistStylus Safety zone around the stylus for pre-positioning	114318
ChannelSettings	
CH_xx	
CfgActivateKinem Active kinematics	204000
kinemToActivate Kinematics to be activated / active kinematics	204001
kinemAtStartup The kinematics to be activated during control start-up	204002
CfgNcPgmBehaviour Specify the behavior of the NC program.	200800
operatingTimeReset Reset the machining time when program starts.	200801
plcSignalCycle PLC signal for the number of the pending machining cycle	200803
CfgGeoTolerance Geometry tolerances	200900
circleDeviation Permissible deviation of the radius	200901
threadTolerance Permissible deviation in successive threads	200902
moveBack Reserve for retraction movements	200903
CfgGeoCycle Configuration of the fixed cycles	201000
pocketOverlap Overlap factor for pocket milling	201001
posAfterContPocket Traverse after machining the contour pocket	201007
displaySpindleErr Display the Spindle is not rotating error message if M3/M4 is not active	201002
displayDepthErr Display the Check the depth sign error message	201003

Depiction in the configuration editor	MP number
apprDepCylWall Behavior when moving to wall of slot in the cylinder surface	201004
mStrobeOrient M function for spindle orientation in machining cycles	201005
suppressPlungeErr Do not show 'Plunging type is not possible' error message	201006
restoreCoolant Behavior of M7 and M8 with Cycles 202 and 204	201008
facMinFeedTurnSMAX Automatic feed rate reduction after attaining SMAX	201009
suppressResMatlWar Do not show "Residual material" warning	201010
CfgStretchFilter Geometry filter for filtering out linear elements	201100
filterType Type of stretch filter	201101
tolerance Maximum distance of the filtered to the unfiltered contour	201102
maxLength Maximum length of the distance resulting from filtering	201103
CfgThreadSpindle Special spindle parameters for threads	113600
sourceOverride Effective override potentiometer for feed rate during thread cutting	113603
thrdWaitingTime Waiting time at reversal point in thread base	113601
thrdPreSwitchTime Advanced switching time of spindle	113602
limitSpindleSpeed Limit of spindle speed with Cycles 17, 207 and 18	113604
CfgEditorSettings Settings for the NC editor	105400
createBackup Generate a backup file *.bak	105401

Depiction in the configuration editor	MP number
deleteBack Behavior of the cursor after deletion of lines	105402
lineBreak Line break on NC blocks with more than one line	105404
stdTNChelp Activate help graphics when entering cycle data	105405
warningAtDEL Confirmation request when deleting an NC block.	105407
maxLineGeoSearch Line number up to which a test of the NC program is to be run.	105408
blockIncrement ISO programming: Block number increment	105409
useProgAxes Specify programmable axes	105410
enableStraightCut Allow or lock paraxial positioning blocks	105411
noParaxMode Hide FUNCTION PARAXCOMP/PARAXMODE	105413
CfgPgmMgt Settings for the file management	122100
dependentFiles Display of dependent files	122101
CfgProgramCheck Settings for tool-usage files	129800
autoCheckTimeOut Timeout for creation of tool-usage files	129803
autoCheckPrg Create tool-usage file for NC program	129801
autoCheckPal Create pallet-usage files	129802
CfgUserPath Paths for the end user	102200
ncDir List of drives and/or directories	102201
fn16DefaultPath Default output path for the FN16: F-PRINT function in the Program Run operating modes	102202
fn16DefaultPathSim Default output path for the FN16: F-PRINT function in the Programming and Test Run operating modes	102203
serialInterfaceRS232	

Depiction in the configuration editor	MP number
CfgSerialPorts Data record belonging to the serial port	106600
activeRs232 Enable the RS-232 interface in the program manager	106601
baudRateLsv2 Data transfer rate for LSV2 communication in baud	106606
CfgSerialInterface Definition of data records for the serial ports	106700
RSxxx	
baudRate Data transfer rate for communication in baud	106701
protocol Communications protocol	106702
dataBits Data bits in each transferred character	106703
parity Type of parity checking	106704
stopBits Number of stop bits	106705
flowControl Type of data-flow checking	106706
fileSystem File system for file operation via serial interface	106707
bccAvoidCtrlChar Avoid control characters in the block check character (BCC)	106708
rtsLow Idle state of the RTS line	106709
noEotAfterEtx Behavior after reception of an ETX control character	106710
Monitoring	
CfgMonUser Monitoring settings for the user	129400
enforceReaction The configured error reactions are enforced	129401
showWarning Display warnings of monitoring tasks	129402
CfgMonMbSection CfgMonMbSection defines monitoring tasks for a certain section of an NC program	02400

Depiction in the configuration editor	MP number
tasks List of monitoring tasks to be performed	133701
CfgMachineInfo General information of the machine operator	131700
machineNickname Custom name (nickname) of the machine	131701
inventoryNumber Inventory number or ID	131702
image Photo or image of the machine	131703
location Machine location	131704
department Department or division	131705
responsibility Responsible for the machine	131706
contactEmail Contact email address	131707
contactPhoneNumber Contact phone number	131708

27.3 User administration roles and rights

27.3.1 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 552

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 ■ HEROS.Printer	■ mnt ■ lp	■ 332 ■ 9

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
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	■ 331
	■ HEROS.ControlFunctions	■ ctrlfct	■ 337
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 334
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 338
	■ HEROS.SetNetwork	■ netadmin	■ 333
	■ HEROS.SetTimezone	■ tz	■ 330
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.LegacyUserNoCtrlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	■ HEROS.ControlFunctions	■ ctrlfct	■ 337
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	■ HEROS.UserAdmin	■ useradmin	■ 336

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	■ NC.OPModeProgramRun	■ NCOpPgmRun	■ 302
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	■ NC.EditNCProgram	■ NCEdNCProg	■ 305
	■ NC.EditPalletTable	■ NCEdPal	■ 309
	■ NC.EditPresetTable	■ NCEdPreset	■ 308
	■ NC.EditToolTable	■ NCEdTool	■ 306
	■ NC.OPModeMDi	■ NCOpMDI	■ 301
	■ NC.OPModeManual	■ NCOpManual	■ 300

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Setter	This role allows you to edit the pocket table.		
	This role grants the rights of the Programmer role, as well as the following rights:		
	■ NC.ApproveFsAxis ■ NC.EditPocketTable ■ NC.SetupDrive ■ NC.SetupProgramRun	■ NCApproveFsAxis ■ NCEdPocket ■ NCSetupDrv ■ NCSetupPgRun	■ 319 ■ 307 ■ 315 ■ 303
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	■ NC.ScheduleProgramRun	■ NCScheduleProgramRun	■ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors. ■ Special functions of Q parameter programming and editing the table header Replacement for code number **555343**		
	■ NC.EditNCProgramAdv ■ 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 tool builder (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 tool builder (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

27.3.2 List of rights

The table below lists all of the individual rights.

Further information: "Rights", Page 552

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operating the machine in the **Manual Operation** and **Electronic handwheel** operating modes.
NC.OPModeMDi	Working in the **Positioning w/ Manual Data Input** operating mode.
NC.OpModeProgramRun	Execution of NC programs in the **Program Run Full Sequence** or **Program run, single block** operating mode.
NC.SetupProgramRun	Probing in the **Manual Operation** and **Electronic handwheel** operating modes. 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
NC.EditNCProgramAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Adjustment of date and time, time zone and time synchronization via NTP and the .
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users

HEROS role name	Description
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMSharedFolders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgramRun	NC program start from an external application (e.g., via the DNC interface)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number **123**
NC.DataAccessServiceRead	Read-only access to the **PLC:** drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through OPC UA NC Server to data defined by the machine manufacturer

27.4 Keycaps for keyboard units and machine operating panels

The keycaps with IDs 12869xx-xx and 1344337-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 361 (FS)

The keycaps with ID 679843-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 360 (FS)

Keycaps for alphabetic keyboard

ID 1286909	-08	-09	-10	-11	-12	-13	-14	-15	-16

ID 1286909	-17	-18	-19	-20	-21	-22	-23	-24	-25

ID 1286909	-26	-27	-28	-29	-30	-31	-32	-33	-34

ID 1286909	-35	-36	–	-38	-39	–	-41	-42	-43
ID 1344337*)	–	–	-01*)	–	–	-02*)	–	–	–

*) With tactile mark

ID 1286909	-44	-45	-46	-47	-48	-49	-50	-51	-52

ID 1286909	-53	-54	-55	-56	-57	-58	-59	-60
ID 679843	–	–	–	-F4	–	–	-F6	–

ID 1286911	-02	-03	-04	-05

ID 1286914 -03

ID 1286915	-02	-03

ID 1286917 -01

Keycaps for operating aids

						
ID 1286909	-61	-62	-63	-64	-65	-66
ID 679843	–	-36	–	–	–	–

Keycaps for operating modes

								
ID 1286909	-67	-68	-69	-70	-71	-72	-73	-74
ID 679843	–	–	-66	–	–	–	–	–

Keycaps for programming

									
ID 1286909	-75	-76	-77	-78	-79	-80	-81	-82	-83

									
ID 1286909	-84	-85	-86	-87	-88	-89	-90	-91	-93

	
ID 1286909	-92
ID 679843	-D6

Keycaps for axis input and value input

	X	Y	Z	A	B	C	U	V	W
	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange
ID 1286909	-94	-95	-96	-4K	-4Y	-4L	-5K	-98	-4Z
ID 679843	-C8	-D3	-53	-54	-C9	-88	-D4	-31	-55

	IV			ESC	INS				DEL
	Orange								
ID 1286909	-97	-0N	-3S	-4S	-4T	-3R	-3T	-3U	-3V
ID 679843	-31	-E2	–	–	–	–	–	–	–

	7	8	9	4	5	1	2	3	0
ID 1286909	-0B	-0C	-0D	-0E	–	-0G	-0H	-2L	-2M
ID 1344337*)	–	–	–	–	-03*)	–	–	–	–

*) With tactile mark

	.	−/+		Q	CE	DEL	NO ENT	END	
ID 1286909	-0K	-0L	-0M	-2N	-0P	-2P	-0R	-0S	-3N

	>>		P	I
			Orange	
ID 1286909	-3W	-3P	-99	-0A

	ENT
ID 1286914	-04

Keycaps for navigation

		HOME	PG UP		GOTO		END	PG DN
ID 1286909	-0T	-0U	-0V	-0W	–	-0Y	-0Z	-1A
ID 1344337*)	–	–	–	–	-04*)	–	–	–

*) With tactile mark

ID 1344337*)	-06	-07
ID 679843	-42	-41

*) With tactile mark

Keycaps for machine functions

									
ID 1286909	-1D	-1E	-1F	-1G	-1H	-1K	-1L	-4X	-1N
ID 679843	-09	-07	-05	-11	-13	-03	-16	-E6	-06
									
ID 1286909	-1P	-1R	-1S	-1T	-1U	-1V	-1W	-1X	-1Y
ID 679843	-10	-14	-23	-22	-24	-29	-02	-21	-20
							 Red	 Green	X–
ID 1286909	-1Z	-2A	-2B	-2C	-2D	-2E	-2H	-2K	-2R
ID 679843	-25	-28	-01	-26	-27	-30	-57	-56	-04
		Z–	V–	+	–				
ID 1286909	–	-2T	-2U	-2Z	-3A	-3E	-3F	-3G	-3H
ID 1344337*)	-05*)	–	–	–	–	–	–	–	–
ID 679843	-15	-08	-12	-59	-60	-40	-73	-76	-74

*) With tactile mark

						C+		C–	
ID 1286909	-3L	-3M	-3X	-3Y	-3Z	-4A	-4B	-4C	-4D
ID 679843	-C6	-75	-46	-47	-F2	-67	-51	-68	-99
	W+	W–	 Red	A+	A–	B+	B–	 Red	 Red
ID 1286909	-4E	-4F	-4H	-4M	-4N	-4P	-4R	-4U	-06
ID 679843	-B8	-B7	-45	-69	-70	-B2	-B1	-52	-18
	 Green	U–	U+			FN 4	FN 5		
ID 1286909	-07	-5A	-5B	-5C	-5D	-4V	-4W	-5E	-5H
ID 679843	-19	-B3	-B4	-61	-62	-A2	-A3	-A4	-E3
									
ID 1286909	-5F	-5G	2Y	-3K	-4G	-2V	-2W	-2X	
ID 679843	-A5	-A6	–	–	–	–	–	–	

									
ID 679843	-43	-44	-B5	-B6	-B9	-C1	-C2	-C3	-C4
									
ID 679843	-C5	-D9	-E1	-92	-91	-93	-94	-63	-64
									
ID 679843	-95	-96	-A1	-C7	-A9	-98	-97	-F3	-72
									
ID 679843	-E4	-E5	-E7	-E8	-48	-49	-50	-65	-17
									
	Green	Green	Green	Red	Red				
ID 679843	-71	-D8	-90	-89	-D7				
									
	Red	Red							
ID 1286909	-2F	-2G							

Other keycaps

									
			Orange	Green	Red				
ID 1286909	-01	-02	-05	-03	-04	–	–	–	–
ID 679843	-33	-34	-35	–	–	-38	-39	-A7	-A8
									
ID 679843	-D5	-F5							

If you need keycaps with additional symbols, please contact HEIDENHAIN.

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: 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

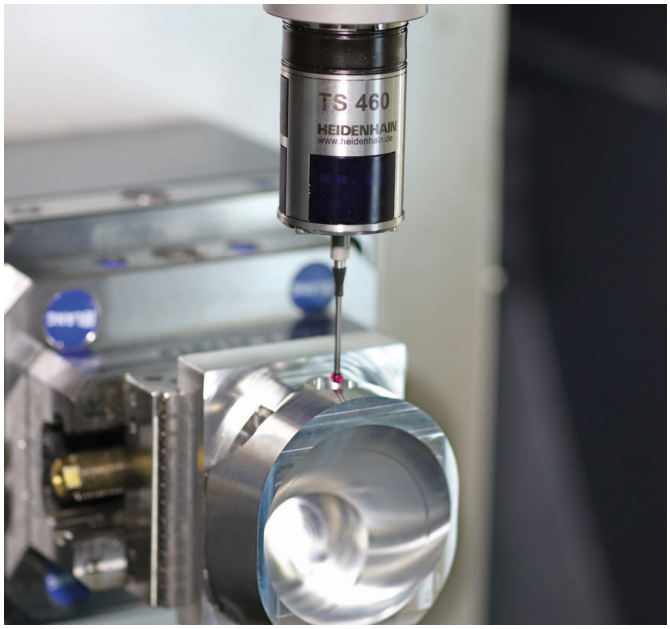
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 150, TS 260, TS 750	Signal transmission by cable
TS 460, TS 760	Radio or infrared transmission
TS 642, TS 740	Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



Tool touch probes

TT 160	Signal transmission by cable
TT 460	Infrared transmission

- Tool measurement
- Wear monitoring
- Tool breakage detection

Original document
1358774-21 · Ver01 · 10/2022 · H · Printed in Germany