HEIDENHAIN

TNC 128

User's Manual
Setup, Testing and Running
NC Programs

NC Software
771841-18

English (en)
10/2023

Controls and displays

Keys

Keys on the screen

Key	Function
⟳	Select screen layout
⟳	Toggle the display between machine operating mode, programming mode, and a third desktop
☐	Soft keys for selecting functions on screen
◁ ▷ △	Switch the soft-key rows

Machine operating modes

Key	Function
🖐	Manual operation
⊛	Electronic handwheel
▤	Positioning with Manual Data Input
▤	Program Run, Single Block
⇥	Program Run, Full Sequence

Programming modes

Key	Function
⇕	Programming
⇥	Test Run

Entering and editing coordinate axes and numbers

Key		Function
X . . .	V	Select the coordinate axes or enter them in the NC program
0 . . .	9	Numbers
.	⁻/₊	Decimal separator / Reverse algebraic sign
P	I	Polar coordinate entry / Incremental values
Q		Q parameter programming / Q parameter status
⊹		Capture actual position
NO ENT		Skip dialog questions, delete words
ENT		Confirm entry and resume dialog
END		Conclude the NC block, end your input
CE		Clear entries or error message
DEL		Abort dialog, delete program section

Tool functions

Key	Function
TOOL DEF	Define tool data in the NC program
TOOL CALL	Call tool data

Managing NC programs and files, control functions

Key		Function
PGM MGT		Select or delete NC programs or files, external data transfer
PGM CALL		Define program call, select datum and point tables
MOD		Select MOD functions
HELP		Display help text for NC error messages, call TNCguide
ERR		Display all current error messages
CALC		Show calculator
SPEC FCT		Show special functions

Navigation keys

Key		Function
↑	→	Position the cursor
GOTO		Go directly to NC blocks, cycles, and parameter functions
HOME		Navigate to the beginning of a program or table
END		Navigate to the end of the program or table row
PG UP		Navigate up one page
PG DN		Navigate down one page
		Select the next tab in forms
		Up/down one dialog box or button

Cycles, subprograms, and program-section repeats

Key		Function
CYCL DEF	CYCL CALL	Define and call cycles
LBL SET	LBL CALL	Enter and call subprograms and program section repeats

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed

Table of contents

1

Basic Information

1.1 About this manual

Safety precautions

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

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

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

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

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

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

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- Escape – Hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

> The information symbol indicates a **tip**.
> A tip provides important additional or supplementary information.

> This symbol prompts you to follow the safety precautions of your machine manufacturer. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

> The book symbol indicates a **cross reference**.
> A cross reference leads to external documentation for example the documentation of your machine manufacturer or other supplier.

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

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

tnc-userdoc@heidenhain.de

HEIDENHAIN | TNC 128 | User's Manual for Setup, Testing and Running NC Programs | 10/2023   **25**

1.2   Control model, software and features

This manual describes functions for setting up the machine as well as for testing and running your NC programs. These functions are provided by controls as of the following NC software numbers.

> **i** 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.

Control model	NC software number
TNC 128	771841-18
TNC 128 Programming Station	771845-18

The machine manufacturer adapts the usable features of the control to his machine by setting appropriate machine parameters. Some of the functions described in this manual may therefore not be among the features provided by the control on your machine tool.

Control functions that may not be available on your machine include:
- Probing functions for the 3D touch probe

In order to find out about the actual features of your machine, please contact the machine manufacturer.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the HEIDENHAIN controls. Participation in one of these courses is recommended to familiarize yourself thoroughly with the control's functions.

Software options

The TNC 128 features various software options, each of which can be enabled separately by your machine manufacturer. The respective options provide the functions listed below:

Additional Axis (option 0 and option 1)

Additional axis	Additional control loops 1 and 2

Touch Probe Functions (option 17)

Touch probe functions	Touch probe cycles:
	■ Set the preset in the **Manual operation** mode of operation
	■ Tools can be measured automatically

HEIDENHAIN DNC (option 18)

	Communication with external PC applications over COM component

Further options available

📖 HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine manufacturer.

For more information, please refer to your machine manufacturer's documentation or the HEIDENHAIN brochure titled **Options and Accessories**.

ID: 827222-xx

📖 **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

Intended place of operation

The control complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

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

To call further information on the control:

▶ Press the **MOD** key

▶ Select the **General Information** group in the MOD menu

▶ Select the **License information** MOD function

When using the OPC UA NC server or DNC server, you can influence the behavior of the control. Therefore, before using these interfaces for productive purposes, verify that the control can still be operated without malfunctions or drops in performance. The manufacturer of the software that uses these communication interfaces is responsible for performing system tests.

New and modified functions with 77184x-18

> 📖 **Overview of new and modified software functions**
>
> Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.
>
> ID: 1322088-xx

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

Further information: User's Manual for **Klartext Programming**

Further information: User's Manual for **Programming of Machining Cycles**

- The **Display Step** software option (option 23) is available in the standard control version. The display step of the axes is no longer limited to four decimal places.

  The machine parameter **displayPace** (no. 101000) allows you to define the display step for the individual axes. The minimum display step of the axes is 0.1 µm or 0.0001°.

- **State Reporting Interface** (software option 137) is no longer available.

New functions

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

Further information: User's Manual for **Klartext Programming**

■ **FUNCTION CORRDATA** allows you to activate a line of the compensation table. The compensation remains effective until the next tool change or the end of the program.

■ **FUNCTION MODE SET** allows you to activate settings defined by the machine manufacturer (e.g., changes to the range of traverse) from within the NC program

■ The function **PRESET SELECT** allows you to activate a preset from the preset table. You can choose to retain active transformations and select the preset to which the function should apply.

■ The function **PRESET COPY** allows you to copy a preset defined in the preset table to another line. You can optionally activate the copied preset and retain the active transformations.

■ The function **PRESET CORR** corrects the active preset.

■ The function **OPEN FILE** allows the control to open different file types (e.g., PNG files) using a suitable additional tool.

- The function **TABDATA** allows you to access the tool table and the compensation tables (*.tco and *.wco) during program run. In order to access the compensation tables, you need to activate them.
  - The function **TABDATA READ** allows you to read a value from a table and save it to a Q, QL, QR, or QS parameter.
  - The function **TABDATA WRITE** allows you to write a value from a Q, QL, QR, or QS parameter into a table.
  - The function **TABDATA ADD** allows you to add a value from a Q, QL, or QR parameter to a value contained in a table.
- The **SELECT FILE** soft key has been added to the selection window provided by the **APPLY FILE NAME** soft key. If the called file is located in the same directory as the file you are calling it from, pressing this soft key loads the name of the file without its path.
- The following NC functions for coordinate transformations have been added:
  - Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes. The **TRANS MIRROR RESET** function allows you to reset mirroring. The NC functions serve as an alternative to Cycle **8 MIRRORING**.
  - The **TRANS SCALE** function lets you change the scale of the contours or distances to the datum, thereby evenly enlarging or shrinking them. This enables you to program shrinkage and oversize allowances, for example. Use the **TRANS SCALE RESET** function to reset scaling. The NC functions serve as an alternative to Cycle **11 SCALING FACTOR**.
  - Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.
- During retraction with **M140 MB MAX**, the control takes into account the safety clearances that can be defined by the machine manufacturer for software limit switches and collision objects. The control reduces the retraction movements by the clearances and stops before the software limit switches.

- In the mask file of the **FN 16: F-PRINT** function, you can define whether the control shows or hides blank lines for undefined QS parameters.
- The function **SYSSTR(ID10321 NR20)** determines the number of the current week in accordance with ISO 8601.
- Using the **SYNTAX** soft key, you can enclose path information in quotation marks in order to use any special characters as part of the path (e.g., **/**). The control provides the **SYNTAX** soft key with the following NC functions:
  - **FN 16: F-PRINT** (ISO: **D16**)
  - **FN 26: TABOPEN** (ISO: **D26**)
  - Cycle **12 PGM CALL** (ISO: **G39**)
  - **CALL PGM** (ISO: **%**)
- The **FN 18: SYSREAD** (ISO: **D18**) functions have been extended:
  - **FN 18: SYSREAD (D18) ID10**: Read program information
    - **NR10**: Counts the number of executions of the current program section
  - **FN 18: SYSREAD (D18) ID15**
    - **NR10**: contents of a Q parameter
    - **NR11**: contents of a QL parameter
    - **NR12**: contents of a QR parameter
  - **FN 18: SYSREAD (D18) ID35 NR2**: active radius compensation
  - **FN 18: SYSREAD (D18) ID50**: values in the tool table
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
  - **FN 18: SYSREAD (D18) ID245 NR1**: Current nominal position of an axis (**IDX**) in the REF system
  - **FN 18: SYSREAD (D18) ID370 NR7**: Reaction of the control if a probing point is not reached during a programmable touch-probe cycle **14xx** (option 17)
  - **FN 18: SYSREAD (D18) ID630**: SIK information of the control
    - **NR3**: SIK generation **SIK1** or **SIK2**
    - **NR4**: Specifies whether and how often a software option (**IDX**) has been enabled on controls with **SIK2**
  - **FN 18: SYSREAD (D18) ID950**: tool-table values for the current tool
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
    - **NR47**: value in the **RN** column
    - **NR48**: value in the **R_TIP** column
  - **FN 18: SYSREAD (D18) ID990 NR28**: Current tool spindle angle
  - **FN 18: SYSREAD (D18) ID1070 NR1**: active feed-rate limit through the **F MAX** soft key
  - **FN 18: SYSREAD (D18) ID10010 NR1** and **NR2**: information about the current main program or the called NC program as a text variable

- **IDX1**: directory path
- **IDX2**: file name
- **IDX3**: file type
- **FN 18: SYSREAD (D18) ID10015**
  - **NR20**: contents of a QS parameter
  - **NR30**: contents of a QS parameter (all characters, except the letters and numbers, are replaced with an underscore (_) character)
- If you use the **SQL EXECUTE** function and the **CREATE TABLE** statement to create a table, then you define the sequence of the columns with the **AS SELECT** statement.
- The **SELECT COMPENS. TABLE** soft key has been added to the soft-key row of the **PGM CALL** functions. This soft key activates the **SEL CORR-TABLE** function that allows you to activate a compensation table for the NC program.

- The control includes the sample tables **WMAT.tab**, **TMAT.tab** and **EXAMPLE.cutd** for automatic cutting data calculation.
- If, after a hardware change or an update, an error occurs when the control is booting, the control will automatically open the error window and display a question-type error. The control displays soft keys providing different response options.
- In the error window, the **ACTIVATE AUTOMATIC SAVING** soft key has been added to **MORE FUNCTIONS**. This soft key allows you to define up to five error numbers. The control will automatically create a service file upon occurrence of these error numbers.
- The control saves active NC programs only up to a maximum size of 10 MB each to a service file. NC programs larger than that are not saved.
- In the optional machine parameter **CfgClearError** (no. 130200), the machine manufacturer defines whether the control automatically clears warning and error messages when an NC program is selected or restarted.
- The CAD Viewer has been enhanced as follows:
  - In **CAD Viewer**, you can choose the **YZ** and **ZX** working planes for milling. You can choose the desired working plane from a selection menu.

- To install or update software version 18, a control with a hard disk size of at least 30 GB is required. The control also needs at least 4 GB RAM.
- The **Test Run** operating mode has been enhanced as follows:
  - In the **Test Run** operating mode, the control uses the active preset.
  - The **RESET THE PRESET** soft key has been added to the **BLANK IN WORK SPACE** menu. For the simulation, you can use this soft key to set the values of the active preset in the principal axis to 0.

  Further information: "Testing and Running", Page 209
- The **OPEN COMPENS. TABLES** soft key has been added to the **Program run, single block** and **Program run, full sequence** operating modes. This soft key allows you to open and edit the active datum table and the active compensation tables.

  Further information: "Compensations during program run", Page 236
- In the **Program run, single block** and **Program run, full sequence** operating modes, the **ACTUAL POSITON CAPTURE** key allows you to load the current position values of an axis into the datum table.
- The control can execute NC programs with the **SECTION MONITORING** NC function. This NC function may be included in NC programs of the TNC7, but has no function on the TNC 128.
- The control supports USB data media with the NTFS file system.
- The control provides the additional tool **Parole** that allows you to open video files.

  Further information: "Software tools for management of external file types", Page 89
- Within the file management, the control hides system files, as well as files and folders whose name begins with a period. If necessary, you can display the files with the **SHOW HIDDEN FILES** soft key.

  Further information: "Additional functions", Page 80

- The general status display has been enhanced as follows:
  - The control displays a corresponding icon in the general status display when tool radius compensation is active.
  - If a feed-rate limit has been activated with the **F MAX** soft key, the control displays an exclamation mark after the feed-rate value in the general status display.

  Further information: "General status display", Page 68
- The TS 760 input option has been added to the **TYPE** column of the touch probe table.
- 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", Page 137

- The following tool types have been added:
  - **Face milling cutter,MILL_FACE**
  - **Chamfer mill, MILL_CHAMFER**
  - **Side milling cutter, MILL_SIDE**

  Further information: "Available tool types", Page 144

- The tool table has been enhanced as follows:
  - In the **RCUTS** column of the tool table, you define the front-face cutting width of a tool (e.g., for indexable inserts).
  - You define the usable length of a tool in the **LU** column of the tool table. The usable length limits the plunging depth of the tool in cycles.
  - You define the neck radius of the tool in the **RN** column of the tool table. This allows the control to display the tool correctly in the simulation (e.g., neck of end mills or side milling cutters).
  - You define a radius at the tip of the tool in the **R_TIP** column of the tool table.
  - 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: "Entering tool data into the table", Page 124

- The **ACTUAL POSITION CAPTURE** soft key allows you to load the actual position of the tool axis into the form view of the tool management as the tool length.

  Further information: "Editing tool management", Page 141

- The **POS. DISP.** soft key enables you to switch the tool table view. The control displays the tool table in combination with the position display or in full screen mode.

- Compensation tables allow you to compensate for tools during program run without having to edit the NC program or the tool tables. The compensation table *.tco is effective in the tool coordinate system as an alternative to the compensation in the tool call.

- The control supports the TS 760 workpiece touch probe.
- A link to the **Firewall settings** HEROS function has been added to the **External access** MOD function.
- A link to the **Certificate and keys** HEROS function has been added to the **External access** MOD function. This function can be used to define settings for secure connections via SSH.

  Further information: "Permitting or restricting external access", Page 278
- If the machine manufacturer has defined the parameter **CfgOemInfo** (no. 131700), then the control displays the **Info about machine manufacturer** area in the **General Information** MOD group.
- The HEROS menu has been enhanced as follows:
  - In the HEROS settings, you can adjust the screen brightness of the control.
  - In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).

  Further information: "Overview of taskbar", Page 289

- The user administration has been enhanced as follows:
  - When user administration is active, the file manager displays the **public** directory that can be accessed by every user.

    When you place the cursor on the **public** directory, the control shows the **ADVANCED ACCESS RIGHTS** soft key. This soft key allows the owner of a file to define the access rights of the following users:
    - Owner
    - Group
    - Other users
  - The **useradmin**, **oem**, and **sys** users can deactivate the user administration.
  - 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 authentication. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.
  - When user administration is active, you can create user-specific private network drive connections. **Single Sign On** allows you to connect to an encrypted network drive while logging on to the control.
  - When configuring the user administration, you can use the **Autologin** function to define a user who will automatically be logged on by the control during booting.

  Further information: "User administration", Page 333
- The optional machine parameter **applyCfgLanguage** (no. 101305) allows you to define whether the HEROS operating system adopts the conversational language defined in machine parameter **ncLanguage** (no. 101301) during booting. If you activate this function, you can change the conversational language only in the machine parameters.
- The optional machine parameter **extendedDiagnosis** (no. 124204) allows you to define whether the control saves graphics journal data after a restart. This data is used to diagnose graphics problems.
- The machine parameter **CfgTTRectStylus** (no. 114300) has been added. This parameter allows you to define settings for a tool touch probe with a cuboid probe contact.

Modified functions

Further information: User's Manual for **Klartext Programming** or **ISO Programming**

Further information: User's Manual for **Klartext Programming**

- To make the control represent the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes and for the radius.
- The pop-up window for tool selection always shows the content of the **NAME** column, even if you are calling the tool with the tool number.
- Within **FUNCTION S-PULSE**, you can define a lower and an upper speed limit for the pulsing speed by means of the **FROM-SPEED** and **TO-SPEED** syntax elements.
- In the **TABDATA WRITE**, **TABDATA ADD** and **FN 27: TABWRITE** (ISO: **D27**) NC functions, you can enter values directly.
- If you program a precision stop of rotary axes using **M134** or **M135**, the control no longer displays an error message. The control ignores these miscellaneous functions.
- The number range for the machine manufacturer's miscellaneous functions has been increased from 1999 to 9999.
- The **FN 10** function also allows you to check QS parameters and texts for inequalities.
- You can use UTF-8 character encoding in the mask file of **FN 16: F-PRINT**.
- The priority of arithmetic operations has been changed in the Q parameter formula.
- You can use combined QS parameters in the **SQL EXECUTE** and **SQL SELECT** functions.

- While program run is interrupted or has been canceled, you can edit Q and QS parameters with numbers 0 to 99, 200 to 1199 and 1400 to 1999 in the **Q parameter list** window.
- Scrolling in the structure window works in the same way as scrolling in the NC program. You can define the position of the active structure block by soft key.
- The control uses the active unit of measure (mm or inches) for calculations in the cutting data calculator.
- The result fields and the diameter field of the cutting data calculator are freely editable.
- The CAD Viewer has been enhanced as follows:
  - **CAD Viewer** performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts 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. If the 2D mode is active, the control shows the Z coordinate dimmed.
  - **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.
  - You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import is not available.
- In the compensation tables (*.tco and *.wco), the input range for all columns containing numerical values has been changed from +/− 999.999 to +/− 999.9999.
- The **FILTER** soft key in the error window was renamed to **GROUPING**. With this soft key, the control groups warnings and error messages.

■ If you set the **MEASURING** soft key to **ON**, then the control displays the following additional information:
  ■ Surface orientation of the current position
  ■ Workpiece number
  ■ Workpiece name
  ■ Note during machining at rapid traverse, thread cutting cycle, or blank form update

  Further information: "Measuring", Page 221

■ The **BLANK IN WORK SPACE** menu provides a soft key that allows you to load the current machine status. The control loads the following information in addition to the active preset:
  ■ Active kinematics
  ■ Active traverse ranges
  ■ Active machining mode
  ■ Active traverse limits

  Further information: "Showing the workpiece blank in the working space ", Page 219

■ The control uses hatch marking to depict threads in the simulation.

■ The simulation takes into account the following columns of the tool table:
  ■ **R_TIP**
  ■ **LU**
  ■ **RN**

■ The control takes into account the following NC functions in the **Test Run** operating mode.
  ■ **FN 27: TABWRITE** (ISO: **D27**)
  ■ **FUNCTION FILE**
  ■ **FUNCTION FEED DWELL**

■ Any display filter you have set in the file manager will be retained even after a control restart.

  Further information: "Selecting drives, directories and files", Page 82

■ If you create a table, and there is at least one prototype available for this file type, the control displays the window **Select table format**. The control also shows whether the prototype is defined in mm or inches. If the control shows both units of measure, you can select a unit of measure.

  The machine manufacturer defines the prototypes. If the prototype contains values, the control transfers these values to the newly created table.

- If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed. If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.
- The machine manufacturer defines the sequence for traversing the axes when returning to the contour.
- The control takes into account manual axes when returning to the contour.
  Further information: "Returning to the contour", Page 247
- In **Program run, single block** operating mode, the control interprets the workpiece blank definition as a single NC block only.
- The control displays the tool index in the block scan pop-up window if needed.
  Further information: "Starting the NC program at any point: Block scan", Page 242
- The control considers the functions **FN 27: TABWRITE** (ISO: D27) and **FUNCTION FILE** only in the operating modes **Program run, single block** and **Program run, full sequence**.
- The additional status display has been enhanced as follows:
  - The control also displays the number of repetitions on the **Overview** and **LBL** tabs of the additional status display after an internal stop.
  - On the **TT** tab of the additional status display, the control displays the tilt angle of the tool touch probe as well as information about the cuboid probe contacts.
  - In **Test Run** operating mode, the control displays the **M** tab of the additional status display when the screen layout **PROGRAM STATUS** is active.
  Further information: "Additional status displays", Page 70
- The handwheel functions have been enhanced as follows:
  - The smallest definable speed level on handwheels with display has been changed from 0.1% to 0.01% of the maximum handwheel speed.
  - 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.
  - When you activate a handwheel with display, the control automatically activates the override potentiometer of the handwheel.
  - In the operating modes **Manual Operation** and **Positioning w/ Manual Data Input**, you can activate a handwheel with display while a macro or a manual tool change is being executed.
- You can activate or deactivate the **F MAX** soft key for reducing the feed rate. The defined value is retained.
  Further information: "Feed-rate limit F MAX", Page 173
- 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", Page 137
- The form view of the tool management shows only those input fields that are needed for the selected tool type.

Further information: "Tool management", Page 139

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

  Further information: "Entering tool data into the table", Page 124

- The control no longer supports the ITC 750 additional operating station.

- If the control is accessed from external, it displays a corresponding icon in the header.

  The control uses an icon to indicate whether a connection configuration is secure or non-secure.

- Limits that have been defined in the **Traverse limits** MOD function are also effective for modulo axes.

  Further information: "Entering traverse limits", Page 276

- **Program run** in the **Machine times** MOD area shows only the times at which at least one axis was moving during program run.

- From within the **Diagnostic functions** MOD group, you can access **TNCdiag** and **Hardware configuration** without a code number.

- The user interface of the **Network settings** window has been changed. For network configuration, use the **Network Connections** window.

  Further information: "Network configuration with Advanced Network Configuration", Page 318

- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

- You can export and import existing network configurations in the **Network settings** window.

  Further information: "The Network settings window", Page 313

- If you enter a password or code number with Caps Lock active, then the control issues a message.
- The machine manufacturer can define a path for saving the values of the QR parameters. If the values are on the **TNC** drive, you can use the **NC/PLC Backup** HEROS function to back up the QR parameters.
  Further information: "Backup and restore", Page 300
- **PKI Admin** now includes the **Advanced settings** tab.
  You can define whether the server certificate should contain static IP addresses and allow connections without an associated CRL file.
- The user administration has been enhanced as follows:
  - When user administration is active, the **Liberating motion** mode requires the NC.OPModeManual permission (i.e., at least the role of **NC.Programmer**).
  - If you use the **Connection to Windows domain** function when configuring the user administration, you can set up a secure connection by activating the **Use LDAPs** check box.
  - If a remote log-in takes place while user administration is inactive, for example via SSH, the control automatically assigns the **HEROS.LegacyUserNoCtrlfct** role.
  - If you deactivate the user administration and select the **Delete existing user databases** check box, the control also deletes the .home folder in the **TNC:** directory.
  - Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.
  - If you have connected the control to the Windows domain, you can export the required configurations for other controls.
  Further information: "User administration", Page 333
- The machine parameter **spindleDisplay** (no. 100807) has been enhanced. The control can also display the spindle position on the **Overview** tab of the additional status display when the spindle is in jog mode.
- The input range of machine parameter **displayPace** (no. 101000) has been extended. The minimum display step of the axes is 0.000001° or mm.
- If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.
  When the control detects a non-secure connection, it displays an informational notice.
- Machine parameter **CfgStretchFilter** (no. 201100) has been removed.

Modified cycle functions with 77184x-18

> 📖 **Overview of new and modified software functions**
>
> Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.
>
> ID: 1322088-xx

- In Cycle **12 PGM CALL** (ISO: G39) you can use the **SYNTAX** soft key to place paths within quotation marks. To separate folders and files within paths, both the **** and the **/** character are permitted.
- Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**) restore the spindle status after machining to that which was active before the cycle.
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.
- Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) check the parameter **Q379 STARTING POINT**. If the value of the starting point is equal to or greater than the value of the parameter **Q201 DEPTH**, then the control issues an error message.
- Parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (**G241**) have been extended. You can define a path for a user macro.
- Cycle **240 CENTERING** (ISO: **G240**) has been expanded in order to consider pre-drilled diameters.

  The following parameters have been added:
  - **Q342 ROUGHING DIAMETER**
  - **Q253 F PRE-POSITIONING**: If parameter **Q342** is defined, feed rate for approaching the deepened starting point
- The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), you program a limit that is perpendicular to the milling direction **Q350**, then the control adds the tool radius to the length of the surface in the unlimited direction. As a result, the control completely machines the defined surface without leaving behind any residual material, as would be caused by the tool radius. If the parameter **Q220** (corner radius) is defined, then the control adds both the tool radius and this value to the length of the surface.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), the parameter **Q389** has been defined with the value 2 or 3 and a lateral limit is defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.
- Cycle **253 SLOT MILLING** monitors a cutting width defined in the column **RCUTS** of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.

- Cycle **251 RECTANGULAR POCKET** takes into consideration a cutting width defined in the column **RCUTS** when calculating the plunging path.
- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.

  The following cycles monitor the usable length LU:
  - All cycles for drilling and boring
  - All cycles for tapping
  - All cycles for the machining of pockets and studs

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**, option 17) allow you to calibrate a tool touch probe with cuboid probe contacts.
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**, option 17) first measures the tool length and then the tool radius.
- Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles.
- Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

2

First Steps

2.1 Overview

This chapter is intended to help you quickly learn to handle the most important procedures on the control. For more information on the respective topic, see the section referred to in the text.

The following topics are included in this chapter:

- Switching on the machine
- Graphically testing the workpiece
- Setting up tools
- Workpiece setup
- Machining the workpiece

> The following topics are covered in the Klartext Programming User's Manual:
> - Switching on the machine
> - Programming the workpiece

2.2 Switching on the machine

Acknowledging the power interruption and moving to the reference points

> **⚠ 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.
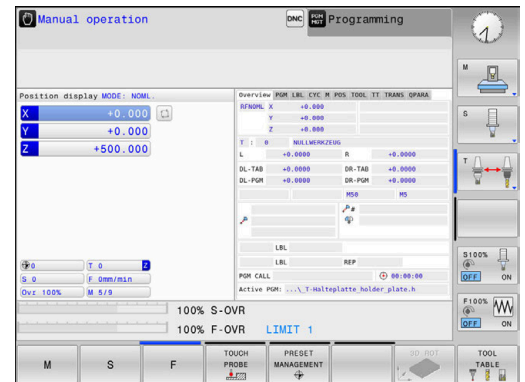
To switch on the machine:

▶ Switch on the power supply for the control and the machine

> The control starts the operating system. This process may take several minutes.

> The control will then display the "Power interrupted" message in the screen header.

CE
▶ Press the **CE** key

> The control compiles the PLC program.

I
▶ Switch on the machine control voltage

> The control checks operation of the emergency stop circuit and goes into Reference Run mode.

▶ Cross the reference point manually in the prescribed sequence: For each axis press the **START** key. If you have absolute linear and angle encoders on your machine there is no need for a reference run

> The control is now ready for operation in the **Manual operation** mode.

Further information on this topic
- Approaching reference points
  Further information: "Switch-on", Page 156
- Operating modes
  Further information: "Programming", Page 66

2.3 Graphically testing the workpiece

Selecting the Test Run operating mode

You can test NC programs in the **Test Run** operating mode:

⇥
- ▶ Press the operating mode key
- > The control switches to the **Test Run** operating mode.

Further information on this topic

- Operating modes of the control
  Further information: "Modes of operation", Page 66
- Testing NC programs
  Further information: "Test run", Page 225
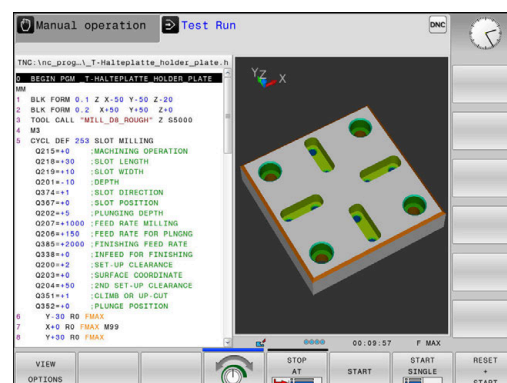
Selecting the tool table

If you have not yet activated a tool table in **Test Run** mode, then you must carry out this step now.

PGM
MGT
- ▶ Press the **PGM MGT** key
- > The control opens the file manager.

SELECT
TYPE
- ▶ Press the **SELECT TYPE** soft key
- > The control shows a soft-key menu for selection of the file type to be displayed.

DEFAULT
- ▶ Press the **DEFAULT** soft key
- > The control shows all saved files in the right-hand window.

←
- ▶ Move the cursor to the left onto the directories

↑
- ▶ Move the cursor to the **TNC:\table** directory

→
- ▶ Move the cursor to the right onto the files

↓
- ▶ Position the cursor on the TOOL.T file (active tool table)

ENT
- ▶ Confirm with the **ENT** key
- > TOOL.T is assigned the status **S** and is thus active for the **Test Run**.

END
- ▶ Press the **END** key to close the file manager

Further information on this topic

- Tool management
  Further information: "Entering tool data into the table", Page 124
- Testing NC programs
  Further information: "Test run", Page 225

Selecting the NC program

PGM MGT	▶ Press the **PGM MGT** key
	> The control opens the file manager.
LAST FILES	▶ Press the **LAST FILES** soft key
	> The control opens a pop-up window with the most recently selected files.
	▶ Use the arrow keys to select the NC program that you want to test.
ENT	▶ Accept with the **ENT** key

Selecting the screen layout and the view

◯	▶ Press the **Screen layout** key
	> The control displays all available alternatives in the soft-key row.
PROGRAM + WORKPIECE	▶ Press the **PROGRAM + WORKPIECE** soft key
	> In the left half of the screen the control shows the NC program; in the right half it shows the workpiece blank.
VIEW OPTIONS	▶ Press the **VIEW OPTIONS** soft key

The control provides the following views:

Soft key	Function
VIEW	Plan view
VIEW	Projection in three planes
VIEW	3D view

Further information on this topic
- Graphic functions
  Further information: "Graphics ", Page 210
- Performing a test run
  Further information: "Test run", Page 225

Starting the test run

RESET
+
START

▶ Press the **RESET + START** soft key
> The control resets the previously active tool data.
> The control simulates the active NC program up to a programmed interruption or to the program end.
▶ While the simulation is running, you can use the soft keys to change views
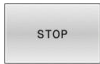
STOP

▶ Press the **STOP** soft key
> The control interrupts the test run.

START

▶ Press the **START** soft key
> The control resumes the test run after an interruption.

Further information on this topic

■ Performing a test run
Further information: "Test run", Page 225
■ Graphic functions
Further information: "Graphics ", Page 210
■ Adjusting the simulation speed
Further information: "Setting the speed of the Test Run mode ", Page 216

2.4 Setting up tools

Selecting the Manual operation mode

Tools are set up in the **Manual operation** mode:

▶ Press the operating mode key
> The control switches to the **Manual operation** mode.

Further information on this topic
■ Operating modes of the control
Further information: "Modes of operation", Page 66
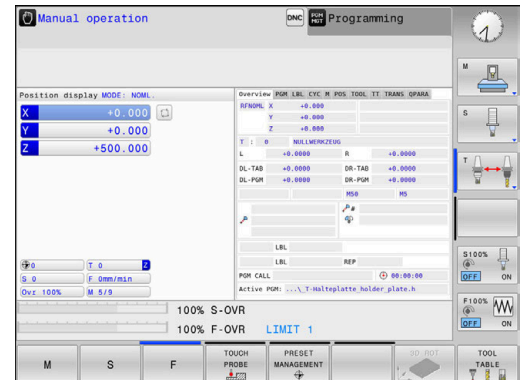
Preparing and measuring tools

▶ Clamp the required tools in their tool holders
▶ When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program
▶ When measuring on the machine: Insert the tool

Editing the TOOL.T tool table

In the TOOL.T tool table (permanently stored under **TNC:\table**), you can save tool data such as length and radius, as well as further tool-specific information that the control needs in order to execute a wide variety of functions.

To enter tool data in the tool table TOOL.T:
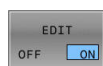
TOOL
TABLE
▶ Press the **TOOL TABLE** soft key

> The control shows the tool table.

EDIT
OFF ON
▶ Set the **EDIT** soft key to **ON**

▶ With the upward or downward arrow keys you can select the tool number that you want to edit

▶ With the rightward or leftward arrow keys you can select the tool data that you want to edit

END
▶ Press the **END** key

> The control closes the tool table and saves the changes.

Further information on this topic

■ Operating modes of the control
Further information: "Modes of operation", Page 66

■ Working with the tool table
Further information: "Entering tool data into the table", Page 124

Editing the TOOL_P.TCH pocket table

⚙ Refer to your machine manual.
The function of the pocket table depends on the machine.

In the pocket table TOOL_P.TCH (permanently saved under **TNC:\table**) you specify which tools your tool magazine contains.

To enter data in the pocket table TOOL_P.TCH:

TOOL
TABLE
▶ Press the **TOOL TABLE** soft key

> The control shows the tool table.

POCKET
TABLE
▶ Press the **POCKET TABLE** soft key

> The control shows the pocket table.

EDIT
OFF ON
▶ Set the **EDIT** soft key to **ON**

▶ With the upward or downward arrow keys you can select the pocket number that you want to edit

▶ With the rightward or leftward arrow keys you can select the data that you want to edit

END
▶ Press the **END** key

Further information on this topic

■ Operating modes of the control
Further information: "Modes of operation", Page 66

■ Working with the pocket table
Further information: "Pocket table for tool changer", Page 129

2.5 Workpiece setup

Selecting the correct operating mode

Workpieces are set up in the **Manual operation** or **Electronic handwheel** operating mode

🖐	▶ Press the operating mode key
	> The control switches to the **Manual operation** mode.

Further information on this topic

■ The **Manual operation** mode
Further information: "Moving the machine axes", Page 160

Clamping the workpiece

Mount the workpiece with a fixture on the machine table so that it is fixed with its edges parallel to the machine axes.

Further information on this topic

■ Presetting with a 3D touch probe
Further information: "Presetting with a 3D touch probe (option number 17)", Page 202

■ Presetting without 3D touch probe
Further information: "Presetting without a 3D touch probe", Page 184

Presetting with a 3D touch probe (option number 17)

Inserting a 3D touch probe

⬛	▶ Select the **Positioning w/ Manual Data Input** operating mode
TOOL CALL	▶ Press the **TOOL CALL** key
	▶ Enter the tool data
ENT	▶ Press the **ENT** key
	▶ Enter the tool axis **Z**
ENT	▶ Press the **ENT** key
END	▶ Press the **END** key
⬛	▶ Press the **NC Start** key

Setting a preset

▶ Select the **Manual operation** mode

▶ Press the **TOUCH PROBE** soft key
> The control displays the available functions in the soft-key row.
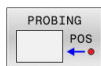
▶ Select the function for setting a preset (e.g., press the **PROBING POS** soft key)

▶ Use the axis direction keys to position the touch probe near the first touch point on the first workpiece edge

▶ Select the probing direction via soft key

▶ Press the **NC Start** key

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

> The control then displays the coordinates of the measured position.

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

> The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point.

▶ Set to 0: Press the **SET PRESET** soft key
▶ Press the **END** soft key to close the menu

▶ Repeat this procedure for all axes, in which you want to set a preset

Further information on this topic

■ Presetting
Further information: "Presetting with a 3D touch probe (option number 17)", Page 202

2.6 Machining the workpiece

Selecting the Program run, single block or Program run, full sequence operating mode

You can run NC programs either in the **Program run, single block** or the **Program run, full sequence** operating mode:

[icon] ▶ Press the operating mode key
> The control switches to the **Program run, single block** operating mode and runs the NC program block by block.
▶ You have to confirm each NC block with the **NC Start** key

[icon] ▶ Press the **Program run, full sequence** key
> The control switches to the **Program run, full sequence** operating mode and runs the NC program, after pressing the NC start key, up to a program interruption or to the end of the program.
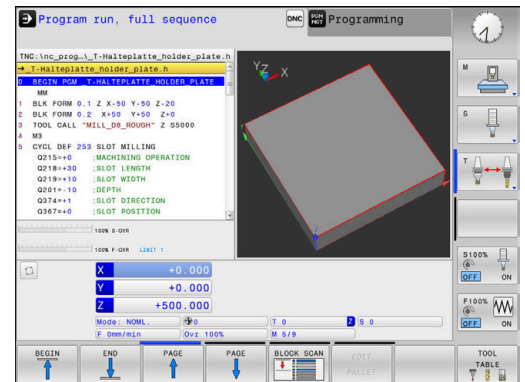
Further information on this topic

■ Operating modes of the control
Further information: "Modes of operation", Page 66

■ Running NC programs
Further information: "Program run", Page 231

Selecting the NC program

PGM
MGT ▶ Press the **PGM MGT** key
> The control opens the file manager.

LAST
FILES ▶ Press the **LAST FILES** soft key
> The control opens a pop-up window with the most recently selected files.
▶ If required, use the arrow keys to select the NC program you want to run. Load with the **ENT** key

Starting an NC program

[icon] ▶ Press the **NC Start** key
> The control runs the active NC program.

Further information on this topic

■ Running NC programs
Further information: "Program run", Page 231

3

Fundamentals

3.1 The TNC 128

The TNC 128 is a workshop-oriented straight-cut control that enables you to program conventional machining operations right at the machine in the easy-to-use Klartext conversational language. It is designed for milling, drilling and boring machines with up to 3 axes. You can also change the angular position of the spindle under program control.

The keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

HEIDENHAIN Klartext

HEIDENHAIN Klartext, the dialog-guided programming language for workshops, is an especially easy method of writing programs. Programming graphics illustrate the individual machining steps for programming the contour. Workpiece machining can be graphically simulated either during a test run or during a program run.

You can also enter and test one NC program while another NC program is machining a workpiece.

Further information: User's Manuals for Klartext Programming

Compatibility

NC programs created on the HEIDENHAIN TNC 124 straight-cut control may not always run on the TNC 128. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

Data security and data protection

The availability of data as well as their guaranteed confidentiality, integrity, and authenticity are decisive for your company's success. HEIDENHAIN therefore places the utmost importance on the protection of relevant data against loss, manipulation, and unauthorized publication.

To ensure that your data is protected actively on the control, HEIDENHAIN offers integrated, state-of-the-art software solutions.

Your control supports the following software solutions:

- **SELinux**
  Further information: "SELinux security software", Page 332
- **Firewall**
  Further information: "Firewall", Page 302
- Integrated browser
  Further information: "Displaying Internet files", Page 92
- Management of external access
  Further information: "Permitting or restricting external access", Page 278
- Monitoring of TCP and UDP ports
  Further information: "Portscan", Page 292
- Remote diagnosis
  Further information: "Secure Remote Access", Page 293
- User administration
  Further information: "User administration", Page 333

These solutions protect the control effectively, but cannot replace company-specific IT security and an integrated overall strategy. In addition to the solutions provided, HEIDENHAIN recommends implementing a company-specific security strategy. This way you ensure effective protection of your data and information, even after they have been exported from the control.

To ensure data security also in the future, HEIDENHAIN recommends informing yourself regularly about product updates and updating the software to the current version.

⚠ 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

---

Virus scanners

Virus scanners can negatively affect the behavior of an NC control.

Possible negative effects are feed rate drops or system crashes, for example. Such negative effects are unacceptable in numerical controls for machine tools. HEIDENHAIN therefore does not offer a virus scanner for the control and does not recommend using a virus scanner.

The control provides the following alternatives:

- **SELinux**
- **Firewall**
- **Sandbox**
- Locking from external access
- Monitoring of TCP and UDP ports

If properly configured, these options ensure extremely effective protection for the data of the control.

If you insist on using a virus scanner, you must operate the control in a protected network (with a gateway and a virus scanner). The subsequent installation of a virus scanner is not possible.

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 12.1-inch screen.

The figure at right shows the keys and controls on the VDU:

1 Header

When the control is on, the screen displays the selected operating modes in the header: The machine operating mode at left and the programming mode at right. The currently active mode is displayed in the larger field of the header, where the dialog prompts and messages also appear.

2 Soft keys

In the footer the control indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The thin bars immediately above the soft-key row indicate the number of soft-key rows that can be called with the keys to the right and left that are used to switch the soft keys. The bar representing the active soft-key row is blue

3 Soft-key selection keys

4 Keys for switching the soft keys

5 Setting the screen layout

6 Key for switchover between machine operating modes, programming modes, and a third desktop

7 Soft-key selection keys for machine manufacturer's

8 Keys for switching the soft keys for machine manufacturer's

9 USB connection
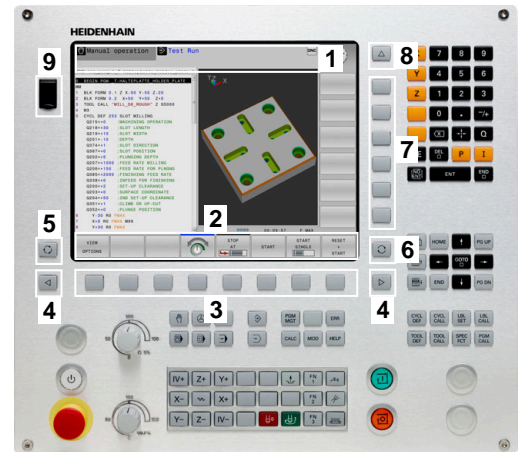
Setting the screen layout

You select the screen layout yourself. In the **Programming** operating mode, for example, you can have the control show the NC program blocks in the left window while the right window displays programming graphics. You could also display the program structure in the right window instead, or display only the NC program blocks in one large window. The available screen windows depend on the selected operating mode.

Setting up the screen layout:

▶ Press the **screen layout** key: The soft-key row shows the available layout options
Further information: "Modes of operation", Page 66

PROGRAM + GRAPHICS

▶ Select the desired screen layout with a soft key

Operating panel

The TNC 128 can be delivered with an integrated operating panel.

1 Machine operating panel
Further information: Machine manual

2 ■ File manager
■ Calculator
■ MOD function
■ HELP function
■ Show error messages
■ Toggle between the operating modes

3 Programming modes

4 Machine operating modes

5 Initiating programming dialogs

6 Navigation keys and **GOTO** jump command

7 Numerical input, axis selection and programming of positioning blocks
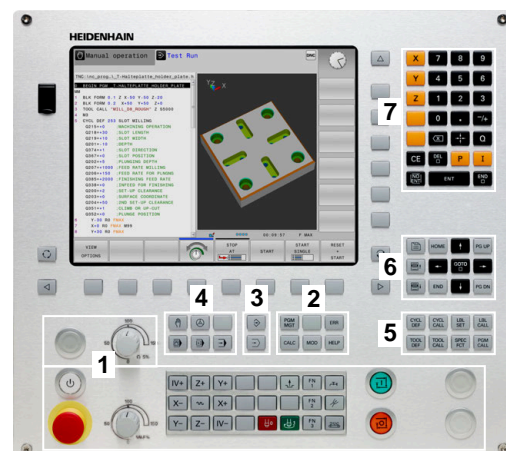
The functions of the individual keys are described on the inside front cover.

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

Switch the control off before cleaning the keyboard unit.

NOTICE
Caution: danger of property damage
Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.
▶ Use permitted cleaning agents only
▶ Use a clean, lint-free cleaning cloth to apply the cleaning agent

The following cleaning agents are permitted for the keyboard unit:
- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:
- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

> **ⓘ** Wear operating gloves to prevent the keyboard unit from getting dirty.

If a trackball is embedded in the keyboard, you need to clean it only when it stops functioning properly.

To clean a trackball (if needed):
- ▶ 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 the cleaning agent onto a cleaning cloth
- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

3.3 Modes of operation

Manual Operation and El. Handwheel

In the **Manual operation** mode of operation, you can set up the machine. You can position the machine axes manually or incrementally, and you can set presets.

The **Electronic handwheel** operating mode supports manual traverse of machine axes with the HR electronic handwheel.

Soft keys for selecting the screen layout

Soft key	Window
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display
POSITION + WORKPIECE	Left: positions, right: workpiece
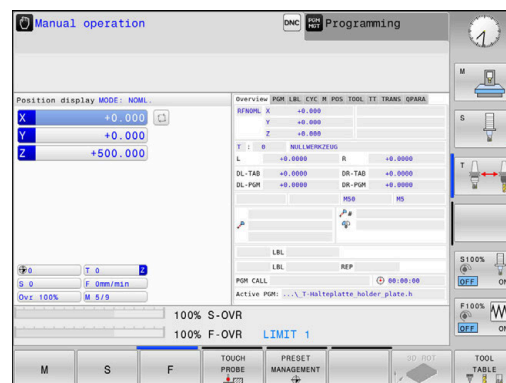
Positioning with Manual Data Input

This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
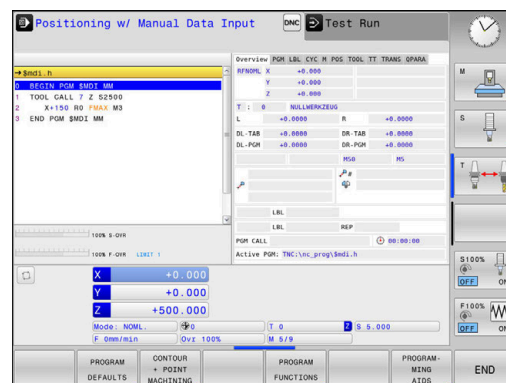
Programming

In this mode of operation you create NC programs. The various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: program structure
PROGRAM + GRAPHICS	Left: NC program, right: programming graphics
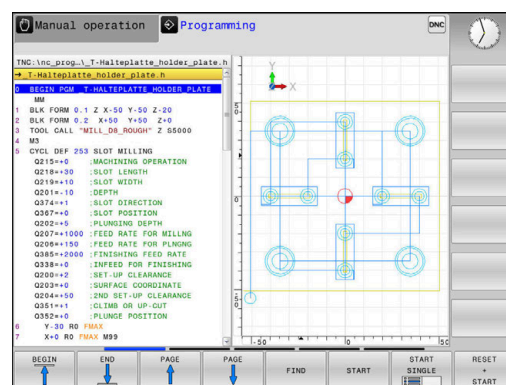
Test Run

In the **Test Run** operating mode, the control simulates NC programs and program sections in order to check them for errors, such as geometrical incompatibilities, missing or incorrect data within the NC program, or violations of the working space. This simulation is supported graphically in different display modes.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece
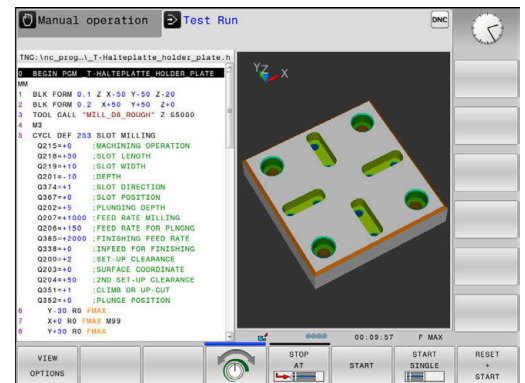
Program Run, Full Sequence and Program Run, Single Block

In the **Program Run Full Sequence** operating mode, the control runs an NC program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the **Program Run Single Block** operating mode, you execute each NC block separately by pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point. The workpiece blank definition will be interpreted as a separate NC block.

Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: structure
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece
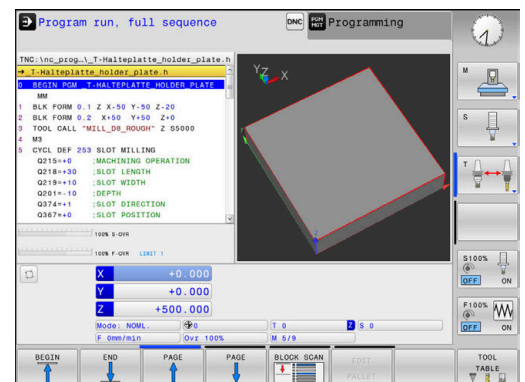
3.4 Status displays

General status display

The general status display in the lower part of the screen informs you of the current state of the machine. The control displays information about axes and positions, as well as technology values and icons for active functions.

The control displays the status in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**

> ℹ️ If the **GRAPHICS** screen layout is selected, the status display is not shown.
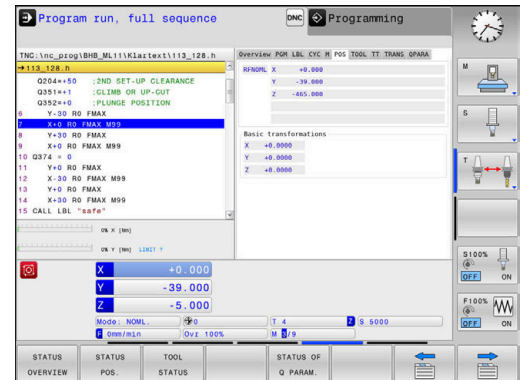
In the **Manual operation** and **Electronic handwheel** operating modes, the control shows the status display in the large window.

Axis display and position display

> ⚙️ Refer to your machine manual.
> The sequence and number of displayed axes is defined by the machine manufacturer.

Icon	Meaning
ACTL	Position display mode (e.g., actual or nominal coordinates of the current position)
	Further information: "Selecting the position display", Page 269
X Y Z	Machine axes
	The selected axis is highlighted
m	The control displays auxiliary axes in lowercase letters
X?	Axis has not been homed
X!	Axis is not in safe operation or it is simulated
┼	Axis is locked
⊕	Axis can be moved with the handwheel

> ℹ️ The machine parameter **CfgPosDisplayPace** (no. 101000) defines the display accuracy by the number of decimal places.

Preset and technology values

Symbol	Meaning
⊕	Number and comment of the active preset from the preset table
	If the preset was set manually, the control displays the text **MAN** behind the symbol
T	Number of the active tool
S	Spindle speed S
F	Feed rate F
	The displayed feed rate in inches corresponds to one tenth of the effective value.
	If a feed-rate limit is active, the control displays an exclamation mark behind the feed rate-value.
	Further information: "Feed-rate limit F MAX", Page 173
M	Active M functions
`M-t/9`	The spindle is controlled from within a cycle (e.g., during tapping)

Icons for active functions

Icon	Meaning
	R+ tool radius compensation is active
	While the **BLOCK SCAN** function is active, the icon is transparent
	R- tool radius compensation is active
	While the **BLOCK SCAN** function is active, the icon is transparent
	Axes are mirrored and moved
	No NC program selected, NC program reselected, NC program aborted via internal stop, or NC program terminated
	In this state, the control has no modally effective program information (i.e., contextual reference), so that all actions are possible (e.g., cursor movements or modification of Q parameters).
	NC program started, execution runs
	For safety reasons, the control permits no actions in this state.
	NC program stopped (e.g., in **Program run, full sequence** operating mode after pressing the **NC Stop** key)
	For safety reasons, the control permits no actions in this state.

Icon	Meaning
	NC program interrupted (e.g., in **Positioning w/ Manual Data Input** operating mode, following the error-free execution of an NC block)
	In this state, the control permits various actions (e.g., cursor movements or the modification of Q parameters). These actions, however, may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in undesired tool positions!
	Further information: "Positioning w/ Manual Data Input operating mode", Page 250 and "Program-controlled interruptions", Page 235
✕	NC program aborted or terminated
S %	The pulsing spindle speed function is active

> ⓘ The order of icons can be changed with the optional **iconPrioList** machine parameter (no. 100813). Only the icon for control-in-operation (STIB) is always visible and cannot be configured.

Additional status displays

The additional status displays contain detailed information on program run. They can be called in all operating modes except for **Programming**. In the **Test Run** operating mode, only a limited status display is available.

Activating the additional status display

◯	▶ Display the soft-key row for screen layout
PROGRAM + STATUS	▶ Select the layout option for the additional status display > In the right half of the screen, the control shows the **Overview** status form.

Selecting an additional status display

▷   ▶ Toggle through the soft-key rows until the **STATUS** soft keys appear
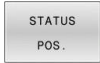
STATUS POS.   ▶ Either select the additional status display directly with the soft key (e.g., positions and coordinates); or

⇥   ▶ Use the toggle soft keys to select the desired view

Select the status displays described below as follows:

- Directly with the corresponding soft key
- Via the toggle soft keys
- Or by using the **next tab** key

⚙ Please note that some of the status information described below is not available unless the associated software option is enabled on your control.

Overview

The control displays the **Overview** status form after startup if you selected the **PROGRAM STATUS** (or **POSITION STATUS**) screen layout. The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

Soft key	Meaning
STATUS OVERVIEW	Position display
	Spindle position
	Depends on the machine parameter **spindleDisplay** (no. 100807)
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program-section repeat
	Name and path of called-up NC program
	Current machining time
	Name and path of the active main program
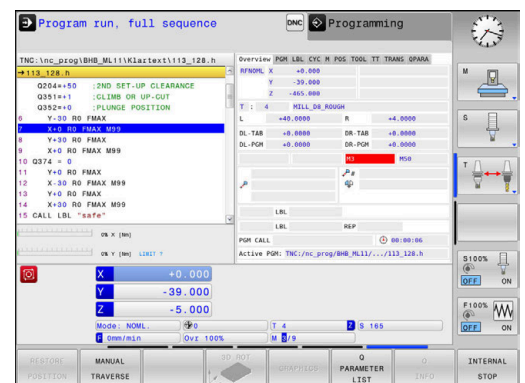
General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name and path of the active main program
	Actual/nominal value counters
	Dwell time counter
	Current machining time
	Called NC programs
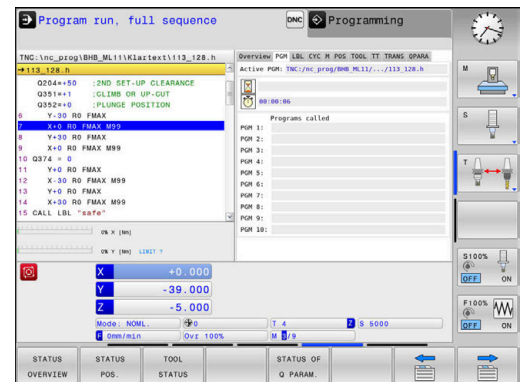
Program-section repeats and subprograms (LBL tab)

Soft key	Meaning
No direct selection possible	Active program-section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
	Active subprograms with number of block in which the subprogram was called and the label number that was called
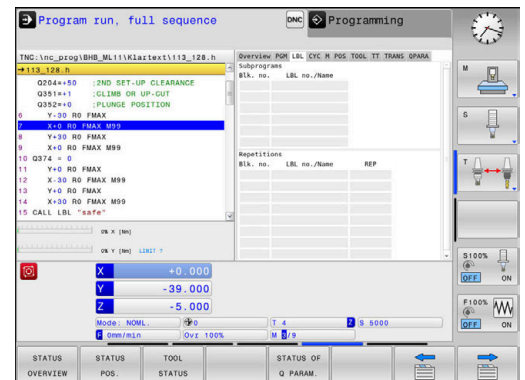
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active machining cycle
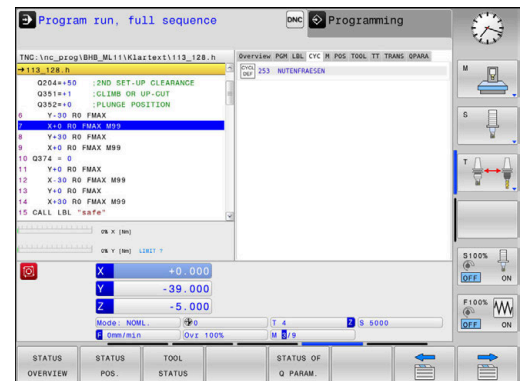
Active M functions (miscellaneous functions, M tab)

Soft key	Meaning
No direct selection possible	List of the active M functions with fixed functionality
	List of the active M functions that have been adapted by the machine manufacturer
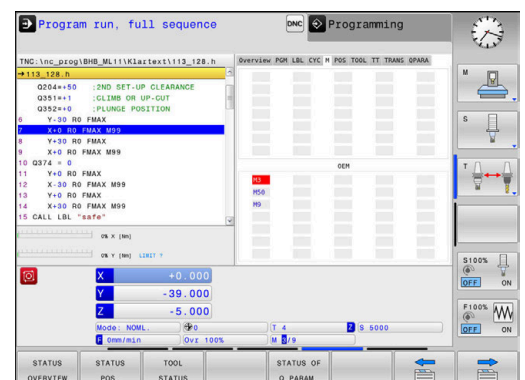
Positions and coordinates (POS tab)

Soft key	Meaning
STATUS POS.	Type of position display (e. g., actual position)
	Axis positions
	Spindle position Depends on the machine parameter **spindleDisplay** (no. 100807)
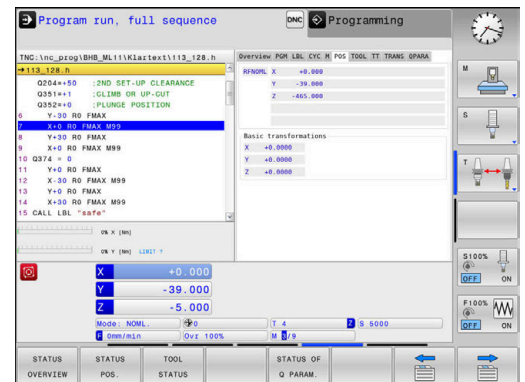
Information on the tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	Display of active tool: ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Finishing allowances (delta values) from the tool table (TAB) and **TOOL CALL** (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for **TOOL CALL** (TIME 2)
	Display of programmed tool and replacement tool
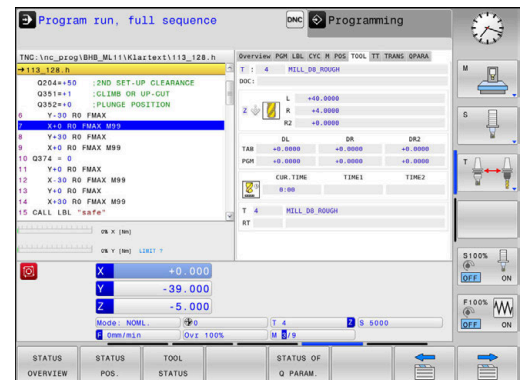
Tool measurement (TT tab)

> ⚙ The control displays this tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Active tool
	Minimum tilt angle (MIN) of the tool touch probe
	Maximum tilt angle (MAX) of the tool touch probe
	Tolerance for tilt angle (DYN)

Measurement results of the cycle:

Field	Meaning
1	Tilt angle in the positive X direction
2	Tilt angle in the positive Y direction
3	Tilt angle in the negative X direction
4	Tilt angle in the negative Y direction
11	X position of tool touch probe in machine coordinate system (M-CS)
12	Y position of tool touch probe in machine coordinate system (M-CS)
13	Z position of tool touch probe in machine coordinate system (M-CS)
14	Diameter or edge length of probe contact
15	Angle of misalignment

> ℹ Your machine manufacturer defines the tolerance for the tilt angle in the optional machine parameter **tippingTolerance** (no. 114319). The control will determine the tilt angle automatically only if a tolerance is defined.
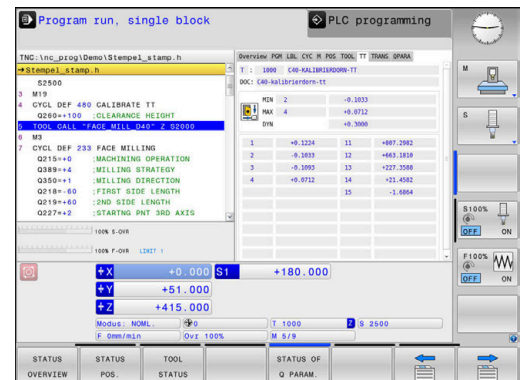
Coordinate transformations

Soft key	Meaning
No direct selection possible	Active transformations
	Name of active datum table, active datum number (**#**), comment from the active row of the active datum table (**DOC**) from Cycle **7**
	Active datum shift (Cycle **7**); the control displays an active datum shift in up to 3 (5) axes
	Active shift in the working plane coordinate system **WPL-CS**
	Mirrored axes (Cycle **8**)
	Active scaling factor (Cycle **11**) / factors (Cycle **26**)
	Scaling datum

Further information: User's Manual for Klartext Programming

Display of Q parameters (QPARA tab)

Soft key	Meaning
STATUS OF Q PARAM.	Display the current values of the defined Q parameters
	Display the character strings of the defined string parameters

> ℹ Press the **Q PARAMETER LIST** soft key. The control opens a pop-up window. For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with commas and connect sequential Q parameters with a hyphen (e.g., 1,3,200-208). The maximum number of characters per parameter type is 132.
>
> The display in the **QPARA** tab always contains eight decimal places. The result of **Q1 = COS 89.999** is shown by the control as 0.00001745, for example. Very large and very small values are displayed by the control in exponential notation. The result of **Q1 = COS 89.999 * 0.001** is shown by the control as +1.74532925e-08, where e-08 corresponds to the factor of 10^{-8}.
>
> The display of QS parameters is restricted to the first 30 characters, i.e. the contents might be truncated.
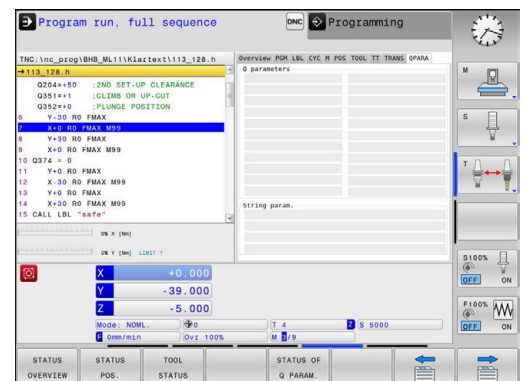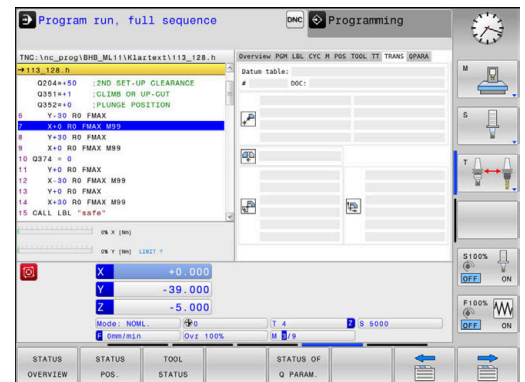
3.5   File management

Files

Files in the control	Type
NC programs	
in HEIDENHAIN format	.H
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (e.g. structure items)	.DEP
Freely definable tables	.TAB
Texts as	
ASCII files	.A
Text files	.TXT
HTML files, e.g. result logs of touch probe cycles	.HTML
Help files	.CHM

When you write an NC program on the control, you must first enter a program name. The control saves the NC program to the internal memory as a file with the same name. The control can also save texts and tables as files.

The control provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

With the control you can manage and save files up to a total size of **2 GB**.

> **i** Depending on the setting, the control generates backup files with the file name extension *.bak after editing and saving NC programs. This reduces the available memory space.

File names

When you store NC programs, tables and texts as files, the control adds an extension to the file name, separated by a point. This extension indicates the file type.

File name	File type
PROG20	.H

File names, drive names and directory names on the control must comply with the following standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (POSIX Standard).

The following characters are permitted:

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 _ -

The following characters have special meanings:

Character	Meaning
.	The last period (dot) in a file name is the extension separator
\ and **/**	Directory separators
:	Separates the drive name from the directory

Do not use any other characters. This helps to prevent file transfer problems, etc.

> **ℹ** 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.

> **ℹ** The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.
>
> **Further information:** "Paths", Page 78

Displaying externally generated files on the control

The control features several software tools which you can use to display the files shown in the table below. Some of the files can also be edited.

File types	Type
PDF files	pdf
Excel tables	xls
	csv
Internet files	html
Text files	txt
	ini
Graphic files	bmp
	gif
	jpg
	png

Further information: "Software tools for management of external file types", Page 89

Directories

To ensure that you can easily find your NC programs and files, we recommend that you organize your internal memory into directories (folders). You can divide a directory into further directories, which are called subdirectories. With the **-/+** key or **ENT** you can show or hide the subdirectories.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash ****.

> The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

The directory AUFTR1 was created on the **TNC** drive. Then, in the AUFTR1 directory, the directory NCPROG was created and the NC program PROG1.H was copied into it. The NC program now has the following path:

TNC:\\AUFTR1\\NCPROG\\PROG1.H

The chart at right illustrates an example of a directory display with different paths.

TNC:\\
— AUFTR1
— NCPROG
— WZTAB
— A35K941
— ZYLM
— TESTPROG
— HUBER
— KAR25T
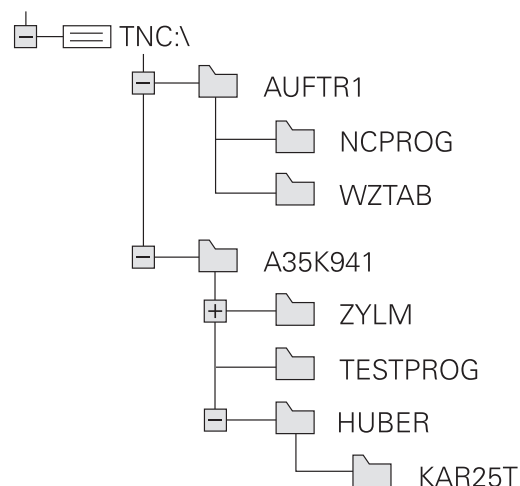
Calling the File Manager

<table>
<tr><td>PGM MGT</td><td>▷ Press the **PGM MGT** key</td></tr>
</table>

> The control displays the file management window (see figure for default setting. If the control displays a different screen layout, press the **WINDOW** soft key).

ℹ️ If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed.

If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.

If you press the **ENT** key, the control always opens an NC program with the cursor on line 0.
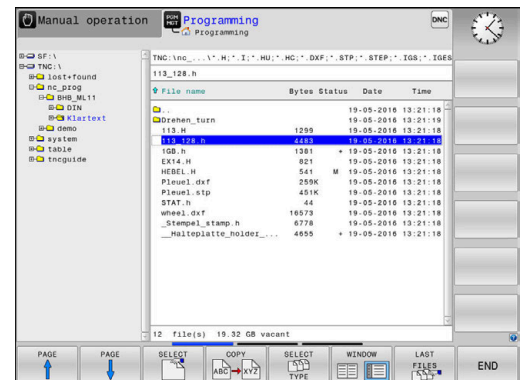
The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive is the internal memory of the control. Other drives are the interfaces (RS232, Ethernet) to which you can connect a PC for example. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. If there are subdirectories, you can show or hide them using the **-/+** key.

If the directory tree is longer than the screen, navigate using the scroll bar or a connected mouse.

The wide window on the right shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

display	Meaning
File name	File name and file type
Bytes	File size in bytes
Status	File properties:
E	The file has been selected in the **Programming** mode of operation
S	File is selected in the **Test Run** operating mode
M	The file is selected in a Program Run mode of operation
+	File has non-displayed dependent files with the extension DEP, e.g. with use of the tool usage test
🔒	File is protected against erasing and editing
🔒	File is protected against deletion and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

To display the dependent files, set the machine parameter **dependentFiles** (no. 122101) to **MANUAL**.

Additional functions

Protecting a file and canceling file protection

▶ Place the cursor on the file you want to protect

| MORE FUNCTIONS | ▶ Select the miscellaneous functions:
Press the **MORE FUNCTIONS** soft key |

| PROTECT | ▶ Activate file protection:
Press the **PROTECT** soft key |
| | ▷ The file is tagged with the "protected" symbol. |

| UNPROTECT | ▶ Cancel file protection:
Press the **UNPROTECT** soft key |

Selecting the editor

▶ Place the cursor on the file you want to open
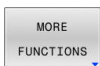
| MORE FUNCTIONS | ▶ Select the additional functions:
Press the **MORE FUNCTIONS** soft key |

| SELECT EDITOR | ▶ Select the editor:
Press the **SELECT EDITOR** soft key |

▶ Mark the desired editor
  - ■ **TEXT EDITOR** for text files (e.g., **.A** or **.TXT**)
  - ■ **PROGRAM EDITOR** for NC programs **.H** and **.I**
  - ■ **TABLE EDITOR** for tables (e.g., **.TAB** or **.T**)

▶ Press the **OK** soft key

Connecting and removing USB storage devices

The control automatically detects connected USB devices with a supported file system.

To remove a USB device:
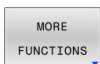
| MORE FUNCTIONS | ▶ Move the cursor to the left-hand window |
| | ▶ Press the **MORE FUNCTIONS** soft key |

| | ▶ Remove the USB device |

Further information: "USB devices on the control", Page 84

ADVANCED ACCESS RIGHTS

The **ADVANCED ACCESS RIGHTS** function can only be used in connection with user administration. This function requires the **public** directory.

Further information: "'public' directory", Page 365

Upon the first activation of user administration, the **public** directory below the **TNC:** drive will be connected.

---

i Access rights can only be defined for files located in the **public** directory.

For all files stored on the **TNC:** drive instead of the **public** directory, the **user** function user will automatically be assigned as the owner.

Further information: "'public' directory", Page 365

Displaying hidden files

The control hides system files, as well as files and folders whose name begins with a period.

NOTICE

Caution: Possible loss of data!

The control's operating system uses certain hidden folders and files. These folders and files are hidden by default. Any manipulation of the system data within the hidden folders might damage the control's software. If you save your own files to these folders, the system will create invalid paths.

▶ Always leave hidden folders and files hidden
▶ Do not use hidden folders and files for saving your own data

---

If required, you can show the hidden files and folders temporarily, e.g., if a file whose name begins with a period is transferred inadvertently.

To show hidden files and folders:
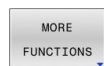
| MORE FUNCTIONS | ▶ Press the **MORE FUNCTIONS** soft key |

| SHOW HIDDEN FILES | ▶ Press the **SHOW HIDDEN FILES** soft key |
| | > The control displays the files and folders. |

Selecting drives, directories and files

| | ▶ Call the file manager by pressing the **PGM MGT** key |

Navigate with a connected mouse or use the arrow keys or the soft keys to move the cursor to the desired position on the screen:

▶ Moves the cursor from the left to the right window, and vice versa

▶ Moves the cursor up and down within a window

▶ Moves the cursor one page up or down within a window

Step 1: Select drive

▶ Move the highlight to the desired drive in the left window

▶ Select a drive: Press the **SELECT** soft key, or

▶ Press the **ENT** key

Step 2: Select a directory

▶ Move the highlight to the desired directory in the left window
> The right-hand window automatically shows all files stored in the highlighted directory

Step 3: Select a file

▶ Press the **SELECT TYPE** soft key

▶ Press the **SHOW ALL** soft key
▶ Move the highlight to the desired file in the right window
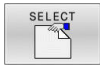
▶ Press the **SELECT** soft key, or

▶ Press the **ENT** key
> The control opens the selected file in the operating mode from which you called the file manager.

ℹ️ If you enter the first letter of the file you are looking for in the file manager, the cursor automatically jumps to the first NC program with the same letter.

Filtering the display

To filter the displayed files:

▶ Press the **SELECT TYPE** soft key

▶ Press the soft key for the desired file type

Alternative:

▶ Press the **SHOW ALL** soft key
> The control displays all files in this folder.

Alternative:

▶ Use wildcards, such as **4*.H**
> The control will show all files of file type .h whose name starts with 4.

Alternative:

▶ Enter the file name extension, e.g. ***.H;*.D**
> The control will show all files of file type .h and .d.

Any display filter you have set will remain effective even after a control restart,

Choosing one of the last files selected

PGM MGT ▸ Call the file manager: Press the **PGM MGT** key.

LAST FILES ▸ Display the last ten files selected: Press the **LAST FILES** soft key

Press the arrow keys to move the cursor to the file you wish to select:

↑ ▸ Moves the cursor up and down within a window

↑
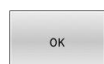
OK ▸ Select the file: Press the **OK** soft key, or

ENT ▸ Press the **ENT** key
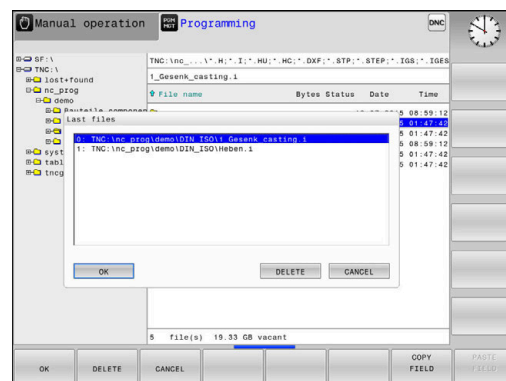
ℹ️ The **COPY FIELD** soft key allows you to copy the path of a marked file. You can reuse the copied path later (e.g., when calling a program with the **PGM CALL** key).

USB devices on the control

ℹ️ Use the USB port only for file transfer and backup. Before editing or running an NC program, save it to the hard disk of the control. This helps to avoid duplicate data maintenance and prevents potential problems resulting from data transfer during program run.

Backing up data from or loading onto the control is especially easy with USB devices. The control supports the following USB block devices:

- Floppy disk drives with FAT/VFAT file system
- Memory sticks with the FAT/VFAT or exFAT file system
- USB sticks with NTFS file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The control automatically detects these USB devices when they are plugged in. If the file system is not supported, the control issues an error message as soon as the stick is plugged in.

> ℹ️ If an error message is displayed when connecting a USB device, check the setting in the **SELinux** security software.
>
> **Further information:** "SELinux security software", Page 332
>
> If the control displays the **USB: TNC does not support device** error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
>
> If the control repeatedly fails to detect a USB device with a file system supported by the control, connect another device to check the port. If this resolves the problem, use the properly working device.

Working with USB devices

> ⚙️ Refer to your machine manual.
>
> Your machine manufacturer can assign permanent names for USB devices.

The USB devices appear as separate drives in the directory tree, so you can use the file management functions described in the earlier chapters.

If a larger file is transferred to a USB device in the file manager, the control displays the **Write access on USB device** dialog until file transfer is completed. The dialog is closed with the **HIDE** soft key and file transfer is continued in the background. The control displays a warning until file transfer is completed.

Removing USB devices

To remove a USB device:
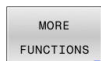
MORE FUNCTIONS	▶ Move the cursor to the left-hand window
	▶ Press the **MORE FUNCTIONS** soft key
[USB icon]	▶ Remove the USB device

Data transfer to or from an external data carrier

> ℹ️ Before you can transfer data to an external data medium,
> you must set up the data interface.
> **Further information:** "Setting up data interfaces",
> Page 305

PGM
MGT
> ▶ Press the **PGM MGT** key

WINDOW
> ▶ Press the **WINDOW** soft key to select the screen
> layout for file transfer

↑
↓
> ▶ Use the arrow keys to move the cursor to the file
> you want to transfer
> \> The control moves the cursor up and down within
> a window.

←
→
> \> The control moves the cursor from the right to the
> left window, and vice versa.
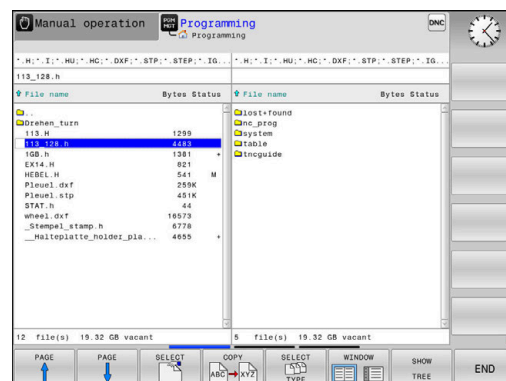
If you wish to copy a file from the control to the external data carrier,
move the cursor in the left window to the file to be transferred.

If you wish to copy from the external data medium to the control,
move the cursor in the right window to the file to be transferred.

SHOW
TREE
> ▶ Press the **SHOW TREE** soft key to select another
> drive or directory
> ▶ Use the arrow keys to select the desired directory

SHOW
FILES
> ▶ Press the **SHOW FILES** soft key
> ▶ Use the arrow keys to select the desired file

COPY
ABC → XYZ
> ▶ Press the **COPY** soft key

ENT
> ▶ Confirm with the **ENT** key
> \> A status window appears on the control,
> informing about the copying progress.

WINDOW
> ▶ Alternative: Press the **WINDOW** soft key
> \> The control displays the standard file manager
> window again.

Preventing incomplete NC programs

The control checks all NC programs for completeness before machining. The control displays a warning if the **END PGM** NC block is missing.

When you start the incomplete NC program in **Program run, single block** or **Program run, full sequence** operating mode, the control aborts with an error message.

Proceed as follows to edit the NC program:

▶ Select the NC program in **Programming** operating mode
> The control opens the NC program and automatically adds the **END PGM** NC block.
▶ Check the NC program, and complete it if necessary

SAVE AS	▶ Press the **SAVE AS** soft key
	> The control saves the NC program with the **END PGM** NC block that has been added.

The control within the network

> ℹ️ Protect your data and the control by running the machines in a secure network.

You connect the control to the network via an Ethernet interface. On the control, you can define general network settings and connect network drives.

Further information: "Ethernet interface ", Page 312

When the control is connected to a network and the file shares are connected, the control displays additional drives in the directory window. The functions for selecting a drive, copying files, etc. also apply to the network drives, provided that the corresponding permission is available.

> ℹ️ The control logs any error messages that occur during network operation.
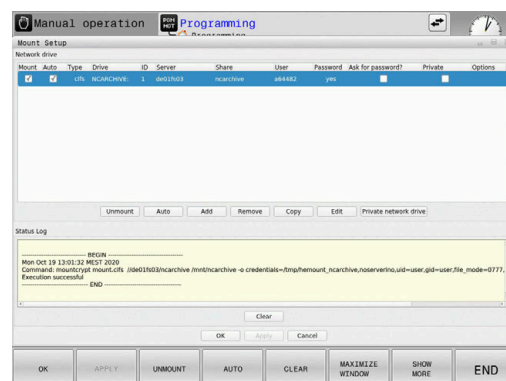
NOTICE

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

Data backup

HEIDENHAIN recommends backing up new NC programs and files created on the control to a PC at regular intervals.

The free-of-charge **TNCremo** software from HEIDENHAIN is a simple and convenient method for backing up data stored on the control.

You can also backup files directly from the control.
Further information: "Backup and restore", Page 300

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Ask your machine manufacturer for assistance, if necessary.

> **ⓘ** 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.

Importing a file from an iTNC 530

> **⚙** Refer to your machine manual.
>
> The machine manufacturer can adapt the
> **ADAPT NC PGM / TABLE** function.
>
> The machine manufacturer can define update rules that
> make it possible, for example, to automatically remove
> umlauts from tables and NC programs.

If you export a file from an iTNC 530 and import it into a TNC 128, depending on the file type you have to adapt its format and content before you can use the file.

The machine manufacturer defines which file types you will be able to import when using the **ADAPT NC PGM / TABLE** function. The control converts the contents of the imported file to a format valid for the TNC 128 and saves the changes to the selected file.

Further information: "Importing tool tables", Page 128

Software tools for management of external file types

The software tools enable you to display or edit various externally created file types on the control.

File types	Description
PDF files (pdf)	Page 90
Excel spreadsheets (xls, csv)	Page 91
Internet files (htm, html)	Page 92
ZIP archives (zip)	Page 94
Text files (ASCII files, e.g. txt, ini)	Page 95
Video files (ogg, oga, ogv, ogx)	Page 95
Graphic files (bmp, gif, jpg, png)	Page 96

> **ℹ** Files with the extensions pdf, xls, zip, bmp, gif, jpg and png must be transmitted in binary format from the PC to the control. Adapt the TNCremo software if required (menu item **Connection** > **Configuring a connection** > **Mode** tab).

Displaying PDF files

To open PDF files directly on the control:

PGM MGT
- ▶ Call the file manager: Press the **PGM MGT** key
- ▶ Select the directory in which the PDF file is saved
- ▶ Move the cursor to the PDF file

ENT
- ▶ Press the **ENT** key
- > The control opens the PDF file in its own application using the **PDF viewer** additional tool.

> **i** With the key combination ALT+TAB, you can always return to the control's user interface while leaving the PDF file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

> **i** If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **PDF viewer** is provided under **Help**.

To exit the **PDF viewer**:
- ▶ Use the mouse to select the **File** menu item
- ▶ Select the **Close** menu item
- > The control returns to the file manager.

If you are not using a mouse, close the **PDF viewer** as follows:

▷
- ▶ Press the key for switching the soft keys
- > The **PDF viewer** opens the **File** select menu.

↓
- ▶ Move the cursor to the **Close** menu item

ENT
- ▶ Press the **ENT** key
- > The control returns to the file manager.

Displaying and editing Excel files

To open and edit Excel files with the extension **xls**, **xlsx** or **csv** directly on the control:

PGM
MGT
- ▶ Call the file manager: Press the **PGM MGT** key
- ▶ Select the directory in which the Excel file is saved
- ▶ Move the cursor to the Excel file

ENT
- ▶ Press the **ENT** key
- > The control opens the Excel file in its own application using the **Gnumeric** additional tool.

> **i** With the key combination ALT+TAB, you can always return to the control's user interface while leaving the Excel file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

> **i** If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use **Gnumeric** is provided under **Help**.

To exit **Gnumeric**:
- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Close**
- > The control returns to the file manager.

If you are not using a mouse, close **Gnumeric** as follows:

▷
- ▶ Press the key for switching the soft keys
- > **Gnumeric** opens the **File** select menu.

↓
- ▶ Move the cursor to the **Close** menu item

ENT
- ▶ Press the **ENT** key
- > The control returns to the file manager.

Displaying Internet files

i   The network must ensure protection against viruses and malicious software. The same applies to access to the Internet or other networks.

The protective measures for this network are the responsibility of the machine manufacturer and the respective network administrator, for example through a firewall.

To open Internet files with the extension **htm** or **html** directly on the control:

PGM
MGT

▶ Call the file manager: Press the **PGM MGT** key
▶ Select the directory in which the Internet file is saved
▶ Move the cursor to the Internet file

ENT

▶ Press the **ENT** key
> The control opens the Internet file in its own application using the **web browser** tool.

i   With the key combination ALT+TAB, you can always return to the control's user interface while leaving the browser open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

i   If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use the **web browser** is provided under **Help**.

After starting the **web browser**, it checks at regular intervals whether updates are available.

You can update the **web browser** only if you deactivate the **SELinux** security software during this period and if a connection with the Internet is available.

i   Reactivate **SELinux** again after the update.

To exit the **web browser**:
- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Quit**
- > The control returns to the file manager.

If you are not using a mouse, close the **web browser** as follows:

| ▷ | ▶ Press the soft-key switchover key: The **web browser** opens the **File** selection menu |

| ↓ | ▶ Move the cursor to the **Quit** menu item |

| ENT | ▶ Press the **ENT** key |
| | > The control returns to the file manager. |

Working with ZIP archives

To open ZIP archives with the extension **zip** directly on the control:

PGM
MGT
▶ Call the file manager: Press the **PGM MGT** key
▶ Select the directory in which the archive file is saved
▶ Move the cursor to the archive file

ENT
▶ Press the **ENT** key
> The control opens the archive file in its own application using **Xarchiver**.

ⓘ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the archive file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

ⓘ If you position the mouse pointer over a button, a brief tool tip explaining the function of this button will be displayed. More information on how to use **Xarchiver** is provided under **Help**.

To exit **Xarchiver**:
▶ Use the mouse to select the **ARCHIVE** menu item
▶ Select the **Exit** menu item
> The control returns to the file manager.

If you are not using a mouse, close **Xarchiver** as follows:

▷
▶ Press the key for switching the soft keys
> **Xarchiver** opens the **ARCHIVE** select menu.

↓
▶ Move the cursor to the **Exit** menu item

ENT
▶ Press the **ENT** key
> The control returns to the file manager.

Displaying or editing text files

Use the internal text editor to open and edit text files (ASCII files, e.g. with the extension **txt**). Proceed as follows:

PGM
MGT
▶ Call the file manager: Press the **PGM MGT** key
▶ Select the drive and the directory in which the text file is saved
▶ Move the cursor to the text file

ENT
▶ Press the **ENT** key
> The control opens the text file with the internal text editor.

ℹ️ Alternatively, you can also open the ASCII files using **Leafpad**. The shortcuts you are familiar with from Windows, which you can use to edit texts quickly (CTRL+C, CTRL+V,...), are available within **Leafpad**.

ℹ️ With the key combination ALT+TAB, you can always return to the control's user interface while leaving the text file open. Alternatively, you can also click the corresponding icon in the task bar to switch back to the user interface of the control.

To open **Leafpad**:
▶ Use the mouse to select the **Menu** HEIDENHAIN icon from the task bar
▶ Select the **Tools** and **Leafpad** menu items in the selection menu

To exit **Leafpad**:
▶ Use the mouse to select the **File** menu item
▶ Select the **Exit** menu item
> The control returns to the file manager.

Displaying video files

⚙️ This function must be enabled and adapted by the machine manufacturer.

To open video files with the extension **ogg**, **oga**, **ogv** or **ogx** directly on the control:

PGM
MGT
▶ Call the file manager: Press the **PGM MGT** key
▶ Select the directory in which the video file is saved
▶ Move the cursor to the video file

ENT
▶ Press the **ENT** key
> The control opens the video file in its own application.

ℹ️ The paid-for Fluendo Codec Pack is required for other formats, such as for MP4 files.

⚙️ Any additional software will be installed by your machine manufacturer.

Displaying graphics files

To open graphics files with the extension **bmp**, **gif**, **jpg** or **png**
directly on the control:

PGM
MGT

▶ Call the file manager: Press the **PGM MGT** key
▶ Select the directory in which the graphics file is
  saved
▶ Move the cursor to the graphics file

ENT

▶ Press the **ENT** key
> The control opens the graphics file in its own
  application using the **Ristretto** software tool.

ⓘ   With the key combination ALT+TAB, you can always return
    to the control's user interface while leaving the graphics file
    open. Alternatively, you can also click the corresponding
    icon in the task bar to switch back to the user interface of
    the control.

ⓘ   More information on how to use **Ristretto** can be found
    under **Help**.

To exit **Ristretto**:

▶ Use the mouse to select the **File** menu item

▶ Select the **Exit** menu item

> The control returns to the file manager.

If you are not using a mouse, close the **Ristretto** software tool as follows:

	▶ Press the key for switching the soft keys
▷	> **Ristretto** opens the **File** selection menu.
↓	▶ Move the cursor to the **Exit** menu item

ENT	▶ Press the **ENT** key
	> The control returns to the file manager.

3.6 Error messages and help system

Error messages

Display of errors

The control displays error messages in the following cases, for example:

- Incorrect input
- Logical errors in the NC program
- Contour elements that are impossible to machine
- Incorrect use of touch probes
- Hardware updates

When an error occurs, the control displays it in the header.

The control uses the following icons and text colors for different error classes:

Icon	Text color	Error class	Meaning
	Red	Error Prompt	The control displays a dialog with several options you can select from. **Further information:** "Detailed error messages", Page 99
	Red	Reset error	The control must be restarted. This message cannot be cleared.
	Red	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Yellow	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.
	Blue	Information	You can continue without clearing the message. You can clear the information at any time.
	Green	Note:	You can continue without clearing the message. The control displays the note until you press the next valid key.

The table rows are ordered by priority. The control displays a message in the header until it is cleared or replaced by a higher-priority message (higher error class).

The control displays long and multi-line error messages in abbreviated form. The complete information on all pending errors is shown in the error window.

An error message that contains an NC block number was caused by an error in the indicated NC block or in the preceding NC block.

Opening the error window

When you open the error window, the complete information on all pending errors will be shown.

ERR
- Press the **ERR** key
- The control opens the error window and displays all accumulated error messages.

Detailed error messages

The control displays possible causes of the error and suggestions for solving the problem:

▶ Open the error window

▶ Position the cursor on the corresponding error message

MORE INFO	▶ Press the **MORE INFO** soft key
	> The control opens a window with information on the error cause and corrective action.
MORE INFO	▶ Exit the info: Press the **MORE INFO** soft key again
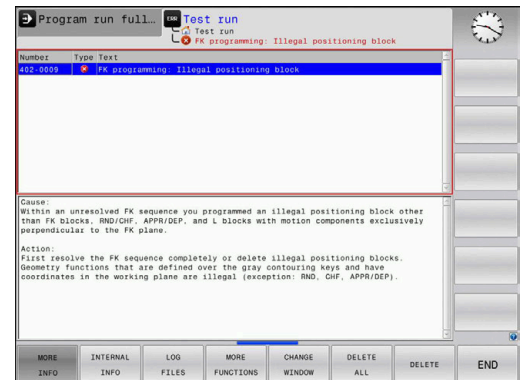
High-priority error messages

When an error message occurs at switch-on of the control due to hardware changes or updates, the control will automatically open the error window. The control displays an error of the question type.

You can correct this error only by pressing the corresponding soft key to acknowledge the question. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.

If a rare **processor check error** should occur, the control will automatically open the error window. You cannot correct such an error.

Proceed as follows:

▶ Shut down the control

▶ Restart

INTERNAL INFO soft key

The **INTERNAL INFO** soft key supplies information on the error message. This information is only required if servicing is needed.

▶ Open the error window

▶ Position the cursor on the corresponding error message

INTERNAL INFO	▶ Press the **INTERNAL INFO** soft key
	> The control opens a window with internal information about the error.
INTERNAL INFO	▶ Exit the detailed information: Press the **INTERNAL INFO** soft key again
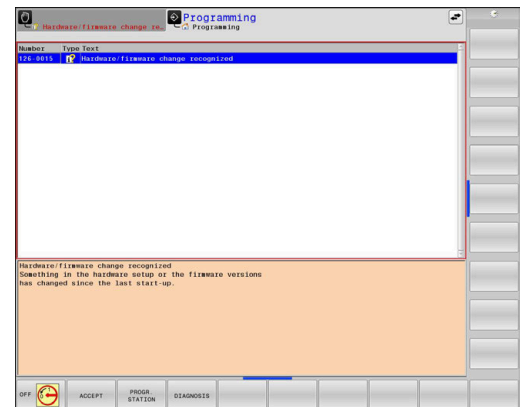
GROUPING soft key

If you activate the **GROUPING** soft key, the control displays all warnings and error messages with the same error number in the same line of the error window. This makes the list of messages shorter and easier to read.

To group the error messages:

| | ▶ Open the error window |
|ERR| |

| | ▶ Press the **MORE FUNCTIONS** soft key |
|MORE FUNCTIONS| |

	▶ Press the **GROUPING** soft key
GROUPING OFF ON	> The control groups identical warnings and error messages.
	> The number of occurrences of the individual messages is indicated in parentheses in the respective line.

| | ▶ Press the **GO BACK** soft key |

ACTIVATE AUTOMATIC SAVING soft key

The **ACTIVATE AUTOMATIC SAVING** soft key allows you to specify error numbers that cause the control to save a service file if an error with that number occurs.

| | ▶ Open the error window |
|ERR| |

| | ▶ Press the **MORE FUNCTIONS** soft key |
|MORE FUNCTIONS| |

	▶ Press the **ACTIVATE AUTOMATIC SAVING** soft key
ACTIVATE AUTOMATIC SAVING	> The control opens the **ACTIVATE AUTOMATIC SAVING** pop-up window.
	▶ Define the entries
	■ **Error number**: Enter the desired error number
	■ **active:** Enable this option to automatically create the service file
	■ **Comment:** Enter a comment on this error number, if required

| | ▶ Press the **STORE** soft key |
|STORE| > If an error with the specified error number occurs, a service file will be saved automatically. |

| | ▶ Press the **GO BACK** soft key |

Deleting errors

The control can automatically clear pending warning or error messages when an NC program is selected or restarted. The machine manufacturer specifies in the optional machine parameter **CfgClearError** (no. 130200) whether these messages will automatically be cleared.

The factory default setting of the control defines that warning and error messages in the **Test Run** and **Programming** operating modes will be cleared automatically from the error window. Messages issued in the machine operating modes will not be cleared.

Clearing errors outside of the error window

CE

▶ Press the **CE** key
> The control clears the errors or notes being displayed in the header.

In certain situations you cannot use the **CE** key for clearing the errors because the key is used for other functions.

Clearing errors

▶ Open the error window
▶ Position the cursor on the corresponding error message

DELETE

▶ Press the **DELETE** soft key

DELETE ALL

▶ As an alternative, clear all errors: Press the **DELETE ALL** soft key

If the cause of the error has not been corrected, the error message cannot be cleared. In this case, the error message remains in the window.

Error log

The control stores errors occurred and important events (e.g., system start) in an error log. The capacity of the error log is limited. When the log is full, the control uses a second file. When this is also full, the first error log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history.

▶ Open the error window

LOG FILES

▶ Press the **LOG FILES** soft key

ERROR LOG

▶ Open the error log file: Press the **ERROR LOG** soft key

PREVIOUS FILE

▶ Set the previous error log if required: Press the **PREVIOUS FILE** soft key

CURRENT FILE

▶ Set the current error log if required: Press the **CURRENT FILE** soft key

The oldest entry is at the beginning of the log file, and the most recent entry is at the end.

Keystroke log

The control stores each key pressed and important events (e.g., system start) in a keystroke log. The capacity of the keystroke log is limited. When the keystroke log is full, the control switches to a second keystroke log. When this is also full, the first keystroke log is deleted and newly written, etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history of the inputs.
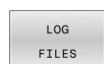
LOG FILES	▶ Press the **LOG FILES** soft key
KEYSTROKE LOG	▶ Open the keystroke log file: Press the **KEYSTROKE LOG** soft key
PREVIOUS FILE	▶ Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key
CURRENT FILE	▶ Set the current keystroke log if required: Press the **CURRENT FILE** soft key
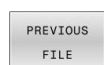
The control saves each key pressed during operation in a keystroke log. The oldest entry is at the beginning, and the most recent entry is at the end of the file.

Overview of the keys and soft keys for viewing the log

Soft key/ Keys	Function
BEGIN	Go to beginning of keystroke log
END	Go to end of keystroke log
FIND	Find text
CURRENT FILE	Current keystroke log
PREVIOUS FILE	Previous keystroke log
↑ ↓	Up/down one line
	Return to main menu

Informational texts

If an operating error occurred, e.g. pressing an impermissible key or entering a value outside of a validity range, the control displays an information text in the header to inform you of the operating error. The control deletes this information text with the next valid entry.

Saving service files

If necessary, you can save the current situation of the control and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke logs as well as other files that contain information about the current status of the machine and the machining).

> ℹ️ In order to facilitate sending service files via email, the control will only save active NC programs with a size of up to 10 MB in the service file. If the NC program is larger, it will not be added to the created service file.

When the same name is entered several times in the **SAVE SERVICE FILES** function, the control saves up to five files and deletes the file with the oldest timestamp, as needed. Make a backup of the service files you created (e.g., by moving them to a different folder).

Saving service files

ERR	▶ Open the error window
LOG FILES	▶ Press the **LOG FILES** soft key
SAVE SERVICE FILES	▶ Press the **SAVE SERVICE FILES** soft key > The control opens a pop-up window in which you can enter a file name or a complete path for the service file.
OK	▶ Press the **OK** soft key > The control saves the service file.

Closing the error window

To close the error window:

END	▶ Press the **END** soft key
ERR	▶ Alternative: Press the **ERR** key > The control closes the error window.

TNCguide: context-sensitive help

Application

ℹ️ Before you can use **TNCguide**, you need to download the help files from the HEIDENHAIN home page.

Further information: "Downloading current help files", Page 109

The **TNCguide** context-sensitive help system contains the user documentation in HTML format. To call **TNCguide**, press the **HELP** key. The control often immediately displays the information specific to the situation in which the help was called (context-sensitive call). If you are editing an NC block and press the **HELP** key, you are usually taken to the exact place in the documentation that describes the corresponding function.

ℹ️ The control tries to start **TNCguide** in the language that you have selected as the user interface language. If the required language version is not available, the control automatically uses the English version.

The following user documentation is available in **TNCguide**:

■ User's Manual for Klartext Programming (**BHBKlartext.chm**)
■ User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
■ List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, in which all existing .chm files are shown in one place.

⚙️ As an option, your machine manufacturer can embed machine-specific documentation in **TNCguide**. These documents then appear as a separate book in the **main.chm** file.
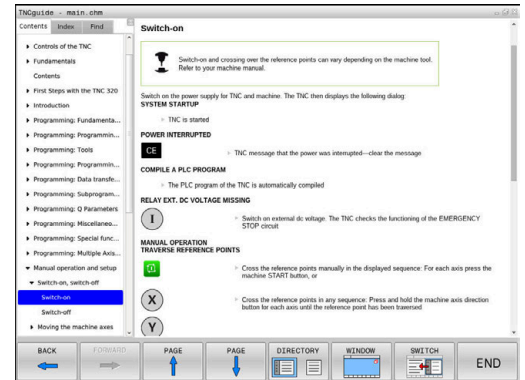
Using TNCguide

Calling TNCguide

You have several options for starting **TNCguide**:

- Use the **HELP** key
- First click the help symbol in the lower right-hand corner of the screen, then click the appropriate soft key
- Open a help file (.chm file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory

> ℹ️ On the Windows programming station, **TNCguide** is opened in the internally defined standard browser.

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse.

Proceed as follows:

▶ Select the soft-key row containing the desired soft key
▶ Click with the mouse on the help symbol that the control displays just above the soft-key row
> The mouse pointer turns into a question mark.
▶ Move the question mark to the soft key for which you want an explanation
> The control opens **TNCguide**. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

Even if you are editing an NC block, context-sensitive help is available:

▶ Select any NC block
▶ Select the desired word
▶ Press the **HELP** key.
> The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.
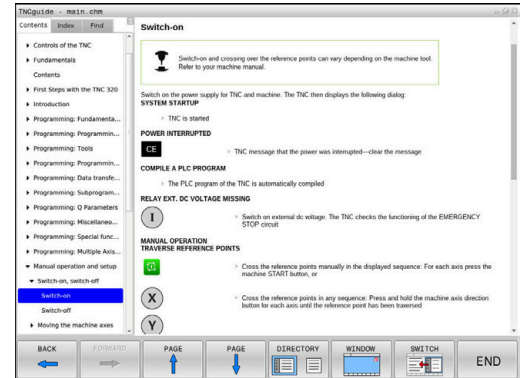
Navigating in TNCguide

It's easiest to use the mouse to navigate in **TNCguide**. A table of contents appears on the left side of the screen. Clicking on the rightward pointing triangle opens subordinate sections, and clicking on the respective entry opens the corresponding page. You can use it in the same way as Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

Soft key/ Keys	Function
↑ ↓	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
→	■ If the table of contents at left is active: Expand the table of contents ■ If the text window at right is active: No function
←	■ If the table of contents at left is active: Collapse the table of contents ■ If the text window at right is active: No function
ENT	■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page
▤	■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the right side of the window ■ If the text window at right is active: Jump back to the left side of the window
▤↕	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to next link
BACK	Select the page last shown
FORWARD	Page forward if you have used the **Select page last shown** function
PAGE ↑	Go back one page
PAGE ↓	Go forward one page

Soft key/ Keys	Function
DIRECTORY	Display or hide table of contents
WINDOW	Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window
SWITCH	The focus is returned to the control application so that you can operate the control while **TNCguide** is open. If the full screen is active, the control reduces the window size automatically before the focus changes
END	Exit **TNCguide**

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the arrow keys.

The left side is active.

▶ Select the **Index** tab
▶ Use the arrow keys or the mouse to select the desired keyword
   Alternative:
▶ Enter the first few characters
▷ The control synchronizes the subject index and creates a list in which you can find the subject more easily.
▶ Use the **ENT** key to call the information on the selected keyword

ⓘ You can enter the search word only with an alphabetic keyboard connected via USB.
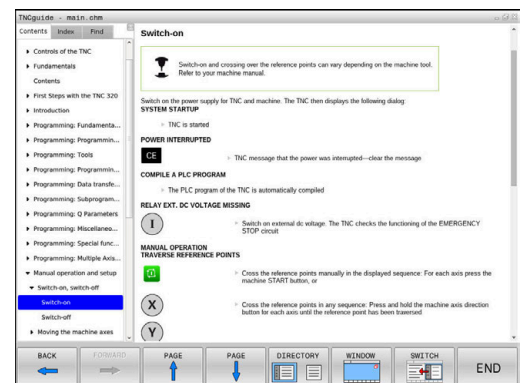
Full-text search

On the **Find** tab, you can search all of **TNCguide** for a specific word.

The left side is active.

▶ Select the **Find** tab

▶ Activate the **Find:** entry field

▶ Enter the search word

▶ Press the **ENT** key

> The control lists all sources containing the word.

▶ Use the arrow keys to navigate to the desired source

▶ Press the **ENT** key to go to the selected source

ℹ The full-text search only works for single words.

If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

You can enter the search word only with an alphabetic keyboard connected via USB.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN website:

http://content.heidenhain.de/doku/tnc_guide/html/en/index.html

Navigate to the suitable help file as follows:

▶ TNC controls

▶ Series (e.g., TNC 100)

▶ Desired NC software number, such as TNC 128 (77184x-18)

> **ℹ** 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.

▶ Select the desired language version from the **TNCguide online help (CHM files)** table

▶ Download the ZIP file

▶ Extract the ZIP file

▶ Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

> **ℹ** When using **TNCremo** to transfer the CHM files to the control, select the binary mode for files with the **.chm** extension.

Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**

Language	TNC directory
Slovenian	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

3.7 NC fundamentals

Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders.

When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. When a reference mark is crossed over, a signal identifying a machine-based reference point is transmitted to the control. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, the machine axes need to move by no more than 20 mm.

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.
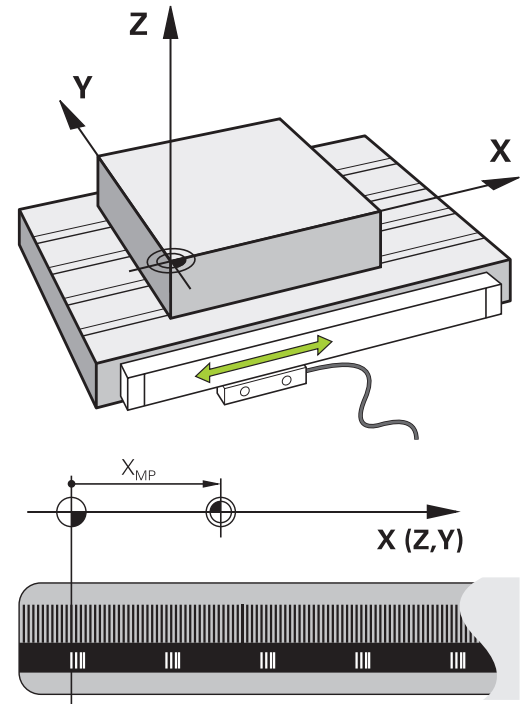
3.8 Accessories: HEIDENHAIN 3D touch probes and electronic handwheels

3D touch probes

Applications for HEIDENHAIN 3D touch probes:

- Quickly and precisely set presets
- Measure the workpiece
- Measure and inspect tools

Touch trigger probes TS 260 and KT 130

The TS 260 and KT 130 touch probes transmit the trigger signals through a cable.

On deflection, a trigger signal is generated, which causes the control to store the current position of the touch probe as the actual value.

Tool touch probe TT 160

The TT 160 touch probe is designed for the efficient and precise measurement and inspection of tool dimensions.

The control offers cycles that enable you to determine the tool length and radius while the spindle is rotating or stationary. The tool touch probe features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf.

A wear-resistant optical switch generates the trigger signal. With the TT 160, signal transmission is by cable.

HR electronic handwheels

Electronic handwheels facilitate manual traversing of the axis slides. A wide range of traverses per handwheel revolution is available. The portable HR 510, HR 520 and HR 550 FS handwheels are avaible from HEIDENHAIN in addition to the HR 130 and HR 150 panel-mounted handwheels.

Further information: "Traverse with electronic handwheels with display", Page 162

4

Tools

4.1 Tool data

Tool number, tool name

Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

> **i** **Permitted characters**: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
>
> The control automatically replaces lowercase letters with corresponding uppercase letters during saving.
>
> **Impermissible characters**: <blank space> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.

Assign unique tool names!

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

- Tool that is in the spindle
- Tool that is in the magazine

> **i** Refer to your machine manual.
>
> If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

- Tool that is defined in the tool table but is currently not in the magazine

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

Database ID

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

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.

Further information: "Tool table fundamentals", Page 119

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

Tool length L

Always enter the tool length **L** as an absolute value based on the tool reference point.

ⓘ The absolute length of the touch probe is always referenced to the tool reference point. The machine manufacturer usually defines the spindle nose as the tool reference point.
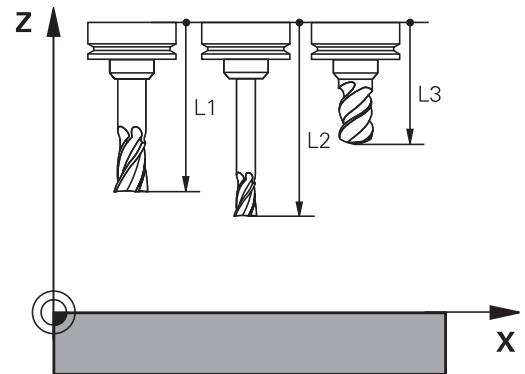
Measuring the tool length

You can measure your tools in the machine (e.g., with a tool touch probe) or externally with a tool presetter. If such measurements are not possible, you can determine the tool length.

You have the following options for determining the tool length:

- With a gauge block
- With a calibration pin (inspection tool)

ⓘ Before you determine tool length, you have to set the preset in the spindle axis.

Determining the tool length with a gauge block

ⓘ You can only set the preset with a gauge block if the tool reference point is at the spindle nose.

Place the preset on the surface you want to touch off with the tool. This surface might have to be created first.

To set the datum with a gauge block:

▶ Place the gauge block on the machine table

▶ Position the spindle nose next to the gauge block

▶ Gradually move in **Z+** direction until you can just slide the gauge block under the spindle nose

▶ Set the preset in **Z**

To determine the tool length:

▶ Insert the tool

▶ Touch off the surface

> The control displays the absolute tool length as the actual position in the position display.
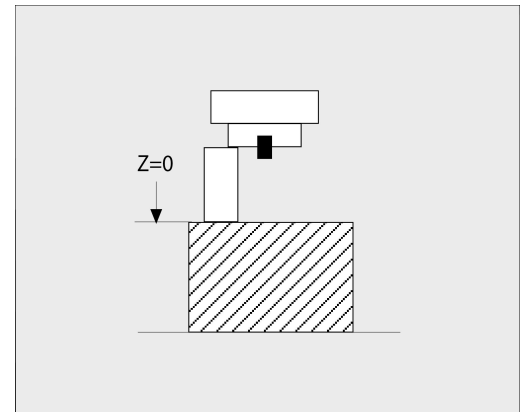
Determining the tool length with a calibration pin and a tool setter

To set the preset with a calibration pin and a tool setter:

▶ Clamp the tool setter onto the machine table.
▶ Bring the flexible inner ring of the tool setter to the same height as the fixed outer ring.
▶ Set the gauge to 0
▶ Move the calibration pin onto the flexible inner ring.
▶ Set the preset in **Z**

To determine the tool length:

▶ Insert the tool
▶ Move the tool onto the flexible inner ring until the gauge displays 0.
> The control displays the absolute tool length as the actual position in the position display.
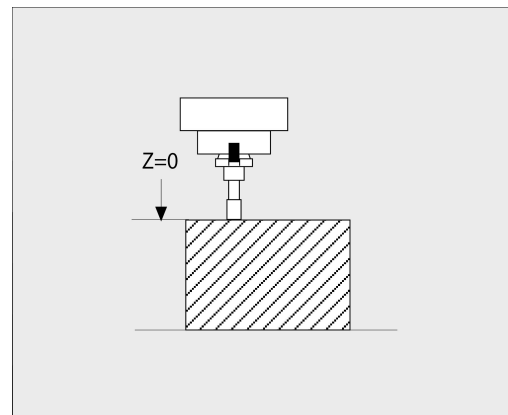
Tool radius R

You can enter the tool radius R directly.

Tool table fundamentals

You can define and store up to 32 767 tools and their tool data in a tool table.

You must use tool tables if:

- you wish to use indexed tools such as stepped drills with more than one length compensation value
  Further information: "Indexed tool", Page 120
- your machine has an automatic tool changer
- you want to work with Cycles **25x**

NOTICE

Caution: Data may be lost!

Deleting line 0 from the tool table will destroy the structure of the table. As a result, locked tools might no longer be recognized as locked and, consequently, the search for a replacement tool will not work, either. The problem cannot be solved by reinserting a line 0. The original tool table will be permanently damaged!

- ▶ Restore the tool table
  - Add a new line 0 to the defective tool table
  - Copy the defective tool table (e.g. toolcopy.t)
  - Delete the defective tool table (current tool.t)
  - Copy the copied tool table (toolcopy.t) as tool.t
  - Delete the copied tool table (toolcopy.t)
- ▶ Contact HEIDENHAIN Service (NC helpline)

ⓘ All table names must start with a letter. Please keep this in mind when creating and managing additional tables.

You can select the table view with the **Screen Layout** key. You can choose between a list view and a form view.

Indexed tool

Step drills, T-slot milling cutters, side milling cutters and, in general, all tools that require the input of multiple length and radius data cannot be fully defined in a single row of the tool table. Each row of the table permits the definition of one length and one radius.

In order to assign multiple compensation data to a tool (multiple tool table rows), add an indexed tool number (e.g., **T 5.1**) to an existing tool definition (**T 5**). Each additional row of the table thus comprises the original tool number, a period and an index (in ascending order from 1 to 9). The original tool table row contains the maximum tool length; the tool lengths in the subsequent table lines are given in descending order of their distance to the tool holder point.

To create an indexed tool number (table row):
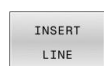
- ▶ Open the tool table
- ▶ Press the **INSERT LINE** soft key
- > The control opens the **Insert line** pop-up window
- ▶ In the **Number of new lines =** input field, enter the number of table rows to add
- ▶ Enter the initial tool number including the index in the **Tool number** entry field
- ▶ Confirm with **OK**
- > The control adds the additional row to the tool table.

Quick search for the tool name:

If the **EDIT** soft key is set to **OFF**, you can search for a tool name. Proceed as follows:

- ▶ Enter the first few characters of the tool name (e.g., **MI**)
- > The control shows a dialog box with the entered text and jumps to the first match.
- ▶ Enter additional characters to narrow down the search result (e.g., **MILL**)
- ▶ If the control cannot find any more matches for the entered search string, you can press the last entered character (e.g., **L**) to jump between matches, as with the cursor keys.

The quick search can also be used for tool selection in the **TOOL CALL** block.

Displaying only specific tool types (filter setting)

▶ Press the **TABLE FILTER** soft key
▶ Select the desired tool type by soft key
> The control displays only tools of the selected type.
▶ Cancel the filter: Press the **SHOW ALL** soft key

> ⚙ Refer to your machine manual.
> The machine manufacturer adapts the features of the filter function to the requirements of your machine.

Soft key	Filter functions of the tool table
TABLE FILTER	Select the filter function
SHOW ALL	Cancel the filter settings and show all tools
DEFAULT FILTER	Use the default filter
DRILL	Show all drills in the tool table
CUTTER	Show all cutters in the tool table
THREADTOOL	Show all taps/thread cutters in the tool table
TCH. PROBE	Show all touch probes in the tool table

Hiding or sorting the tool table columns

You can adapt the layout of the tool table to your needs. Columns that are not to be displayed can be simply hidden:

▶ Press the **SORT/ HIDE COLUMNS** soft key
▶ Select the appropriate column name with the arrow key
▶ Press the **HIDE COLUMN** soft key to remove this column from the table view

You can also modify the sequence of columns in the table:

▶ You can also modify the sequence of columns in the table with the **Move before:** dialog. The entry highlighted in **Displayed columns:** is moved in front of this column

You can use a connected mouse or the navigation keys to move through the form.

Proceed as follows:

▶ Press the navigation keys to go to the input fields
▶ Use the arrow keys to navigate within an input field
▶ Press the **GOTO** key to open the selection menu

> ℹ The function **freeze number of columns** enables you to determine how many columns (0-3) the control will freeze to the left border of the screen. These columns will remain visible when you navigate to the right within the table.

Creating and activating an INCH-based tool table

ⓘ If you choose to switch the units for your control to **INCH**, the tool table unit of measure will not be adjusted automatically.

If you want to change the unit of measure in the table as well, you need to create a new tool table.

To create and activate an **INCH**-based tool table:

▶ Select the **Positioning w/ Manual Data Input** operating mode
▶ Call the zero tool (T0)
▶ Restart the control
▶ Do **not** acknowledge **Power interrupted** with **CE**
▶ Select the **Programming** operating mode

▶ Open the file manager
▶ Open the **TNC:\table** folder
▶ Rename the **tool.t** file, e.g. to **tool_mm.t**
▶ Create a new file and name it **tool.t**
▶ Select the **INCH** unit of measure
> The control opens the new, empty tool table.
▶ Append lines, e.g. 100 lines
> The control adds the lines
▶ Position the cursor in column **L** of line **0**
▶ Enter **0**
▶ Position the cursor in column **R** of line **0**
▶ Enter **0**
▶ Confirm your input
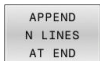
▶ Open the file manager
▶ Open any NC program
▶ Select **Manual operation** mode
▶ Acknowledge **Power interrupted** with **CE**
▶ Open the tool table
▶ Review the tool table

ⓘ The preset table is another table whose unit of measure is not adjusted automatically.

Further information: "Creating and activating an INCH-based preset table", Page 175

Entering tool data into the table

Standard tool data

Parameter	Meaning	Dialog
T	Number by which the tool is called in the NC program (e.g.: 5, indexed: 5.2)	-
NAME	Name by which the tool is called in the NC program (max. 32 characters, all capitals, no spaces)	**Tool name?**
L	Tool length L	**Tool length?**
R	Tool radius R	**Tool radius?**
R2	Tool radius R2 for toroid cutters (only for graphical representation of a machining operation with Ball-nose cutter)	**Tool radius 2?**
DL	Delta value for tool length L	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize?**
DR2	Delta value for tool radius R2	**Tool radius oversize 2?**
TL	Set tool lock (**TL**: for **T**ool**L**ocked)	**Tool locked? Yes=ENT/No=NOENT**
RT	Number of a replacement tool (**RT** for **R**eplacement **T**ool) An empty field or input **0** means there is no replacement tool	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	**Maximum tool age?**
TIME2	Maximum tool life in minutes during a tool call: If the current tool age reaches or exceeds this value, the control inserts the replacement tool during the next **TOOL CALL** (with the tool axis specified)	**Max. tool age for TOOL CALL?**
CUR_TIME	The current tool age equals the time during which the tool is cutting a workpiece. The tool is cutting a workpiece when the spindle is switched on and the control moves the tool at the machining feed rate. The control counts this time automatically and enters the current tool age in minutes. You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring. The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.	**Current tool age?**
TYP	Tool type: Press the **ENT** key to edit the field. The **GOTO** key opens a window in which you can select the tool type. In the tool management, use the **SELECT** soft key to open the pop-up window. You can assign tool types to specify the display filter settings such that only the selected type is visible in the table	**Tool type?**
DOC	Comment on tool (max. 32 characters)	**Tool description**
PLC	Information on this tool that is to be sent to the PLC	**PLC status?**
LCUTS	Tooth length of the tool	**Tooth length in the tool axis?**

Parameter	Meaning	Dialog
LU	Usable length of the tool By entering a value here, you can limit the tool's plunging depth in the cycles. If used in conjunction with **RN**, then **LU** may also be greater than **LCUTS**.	**Usable length of the tool?**
RN	Neck radius for the exact definition of the tool for graphic simulation (e.g., necks of end mills or side milling cutters) A neck radius **RN** that is shown in the graphic simulation is possible only if **LU** > **LCUTS**.	**Neck radius of the tool?**
TMAT	Cutting material of the tool for the cutting data calculator	**Tool material?**
CUTDATA	Cutting data table for the cutting data calculator	**Cutting data table?**
NMAX	Limit the spindle speed for this tool. The programmed value is monitored (error message) as well as an increase in the shaft speed via the potentiometer. Function inactive: Enter **-**. **Input range**: 0 to +999 999; if function not active: enter **-**	**Maximum speed [rpm]**
TP_NO	Reference to the number of the touch probe in the touch-probe table	**Number of the touch probe**
T-ANGLE	Point angle of the tool.	**Point angle**
PITCH	Thread pitch of the tool. Is used by Cycles **206**, **207**, and **208**. A positive algebraic sign means a right-hand thread.	**Tool thread pitch?**
LAST_USE	The time at which the tool was last used The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program.	**Date/time of last tool usage**
KINEMATIC	Use the **SELECT** soft key to show the tool carrier kinematics. In the tool management, use the **SELECT** soft key and the **OK** soft key to adopt the file name and the path **Further information:** "Assigning a tool carrier", Page 154	**Tool-carrier kinematics**
OVRTIME	Time for exceeding the tool life in minutes **Further information:** "Overtime for tool life", Page 133 Refer to your machine manual. Function is defined by the machine manufacturer!	**Tool life expired**
RCUTS	Front-face cutting width of the tool (e.g., for indexable inserts)	**Width of the indexable insert**
DB_ID	The database-ID allows you to identify a tool (e.g., within a tool management system by using client applications). **Further information:** "Database ID", Page 116 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.	**ID for central tool management**
R_TIP	Radius at the tool tip for exact definition of the tool for graphical simulation and automatic calculation within cycles for tools such as countersinks.	**Radius at the tip**

Tool data for automatic tool measurement

> ⚙ Refer to your machine manual.
>
> The machine manufacturer defines whether the **R-OFFS** offset will be taken into account for a tool with **CUT** 0.
>
> Your machine manufacturer defines the default value for the **R-OFFS** and **L-OFFS** columns.

Parameter	Meaning	Dialog
CUT	Number of teeth (99 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 5.0000 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 5.0000 mm	**Wear tolerance: radius?**
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: Radius 2?**
DIRECT	Cutting direction of the tool for measuring the tool during rotation	**Cutting direction? M4=ENT/M3=NOENT**
R-OFFS	Length measurement: Offset between the center of the tool touch probe and the tool center	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper surface of tool touch probe and tool tip in addition to **offsetToolAxis**	**Tool offset: length?**
LBREAK	Permissible deviation from tool length **L** for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 9.0000 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 9.0000 mm	**Breakage tolerance: radius?**

Editing the tool tables

The tool table that is active during execution of the part program is designated TOOL.T and must be saved in the **TNC:\table** directory.

Other tool tables that are to be archived or used for test runs are given different file names with the extension .T. By default, for the **Test Run** and **Programming** operating modes, the control also uses the TOOL.T tool table. In the **Test Run** operating mode, press the **TOOL TABLE** soft key to edit it.

To open the tool table TOOL.T:

▶ Select any machine operating mode

▶ Select the tool table: Press the **TOOL TABLE** soft key
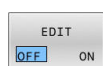
▶ Set the **EDIT** soft key to **ON**
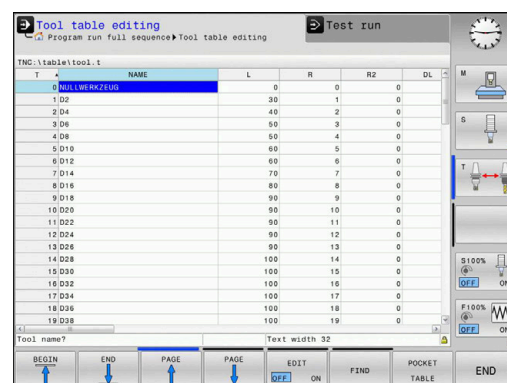
> **ℹ** If you edit the tool table, the selected tool is locked. If this tool is required in the NC program being used, the control shows the message: **Tool table locked**.
>
> If a new tool is created, the length and radius columns remain empty until entered manually. An attempt to insert such a newly created tool will be aborted by the control and an error message will appear. This means you cannot insert a tool for which no geometry data are available yet.

To use the alphabetic keyboard or a connected mouse for navigation and editing:

- Arrow keys: move from one cell to the next
- ENT key: jump to the next cell; with selection fields: open the selection dialog
- Click a cell: move to the cell
- Double-click a cell: place the cursor in the cell; with selection fields: open the selection dialog

Soft key	Editing functions of the tool table
BEGIN	Select the beginning of the table
END	Select the end of the table
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
FIND	Find the text or number
BEGIN LINE	Go to beginning of line
END LINE	Go to end of line
COPY FIELD	Copy active field
PASTE FIELD	Insert copied field
APPEND N LINES AT END	Add the entered number of lines (tools) at the end of the table
INSERT LINE	Insert a line with definable tool number
DELETE LINE	Delete the current line (tool)
SORT	Sort the tools according to the content of a column

Soft key	Editing functions of the tool table
SELECT	Select possible entries from a pop-up window
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell

Importing tool tables

> ⚙ Refer to your machine manual.
>
> The machine manufacturer can adapt the
> **ADAPT NC PGM / TABLE** function.
>
> The machine manufacturer can define update rules that
> make it possible, for example, to automatically remove
> umlauts from tables and NC programs.

If you export a tool table from an iTNC 530 and import it into a
TNC 128, you have to adapt its format and content before you can
use the tool table. On the TNC 128, you can adapt the tool table
conveniently with the **ADAPT NC PGM / TABLE** function. The control
converts the contents of the imported tool table to a format valid for
the TNC 128 and saves the changes to the selected file.

Proceed as follows:

▶ Save the tool table of the iTNC 530 to the **TNC:\table** directory

⮀ ▶ Select the **Programming** operating mode

[PGM MGT] ▶ Press the **PGM MGT** key

[↑] ▶ Move the cursor to the tool table you want to import

[MORE FUNCTIONS] ▶ Press the **MORE FUNCTIONS** soft key
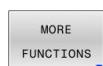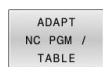
[ADAPT NC PGM / TABLE] ▶ Press the **ADAPT NC PGM / TABLE** soft key

> The control asks you whether you want to overwrite the selected tool table.

▶ Press the **CANCEL** soft key

▶ Alternative: Press the **OK** soft key to overwrite

▶ Open the converted table and check its contents

> New columns in the tool table are highlighted green.

▶ Press the **REMOVE UPDATE INFO** soft key

> The green columns are displayed in white again.

ⓘ The following characters are permitted in the **Name** column of the tool table: # $ % & , - . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z _
During the import, commas will be converted to periods.

The control overwrites the active tool table when importing an external table with the same name. To prevent data loss, back up the original tool table before you start the import!

When iTNC 530 tool tables are imported, all defined tool types are imported as well. Tool types not present are imported as type **Undefined**. Check the tool table after the import.

Pocket table for tool changer

⚙ Refer to your machine manual.
The machine manufacturer adapts the features of the pocket table to the requirements of your machine.

You need a pocket table for automatic tool changing. You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:\table** directory. The machine manufacturer can amend the name, path, and content of the pocket table. If you wish, you can select different views using the soft keys in the **TABLE FILTER** menu.
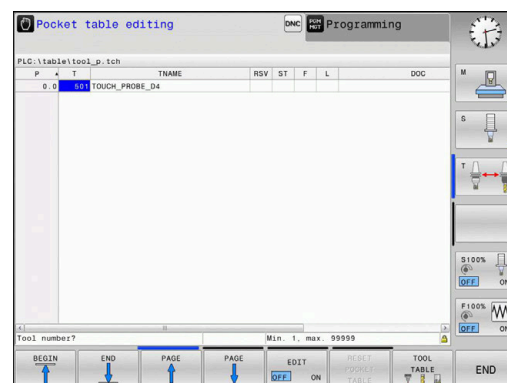
Editing a pocket table in a Program Run operating mode

▶ Select the tool table: Press the **TOOL TABLE** soft key

▶ Press the **POCKET TABLE** soft key

▶ If required, set the **EDIT** soft key to **ON**

Selecting a pocket table in Programming mode

To select the pocket table in the Programming operating mode:

PGM MGT	▶ Call the file manager: Press the **PGM MGT** key. ▶ Press the **SHOW ALL** soft key ▶ Select a file or enter a new file name ▶ Confirm your entry with the **ENT** key or the **SELECT** soft key

Parameter	Meaning	Dialog
P	Pocket number of the tool in the tool magazine	-
T	Tool number	**Tool number?**
RSV	Pocket reservation for box magazines	**Pocket reserv.: Yes = ENT / No = NOENT**
ST	Special tool (**ST**); If your special tool blocks pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	**Special tool?**
F	The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (**L**: for **L**ocked)	**Pocket locked Yes = ENT / No = NO ENT**
DOC	Display of the comment to the tool from TOOL.T	-
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
P1 ... P5	Function is defined by the machine manufacturer. The machine tool documentation provides further information	**Value?**
PTYP	Tool type. Function is defined by the machine manufacturer. The machine tool documentation provides further information	**Tool type for pocket table?**
LOCKED_ABOVE	Box magazine: Lock the pocket above	**Lock the pocket above?**
LOCKED_BELOW	Box magazine: Lock the pocket below	**Lock the pocket below?**
LOCKED_LEFT	Box magazine: Lock the pocket at left	**Lock the pocket at left?**
LOCKED_RIGHT	Box magazine: Lock the pocket at right	**Lock the pocket at right?**

Soft key	Editing functions for pocket tables
BEGIN ↑	Select the table start
END ↓	Select the table end
PAGE ↑	Select the previous page in the table
PAGE ↓	Select the next page in the table
RESET POCKET TABLE	Reset pocket table Depends on optional machine parameter **enableReset** (no.106102) ℹ️ When using extended tool management, you can reset the pocket table only before acknowledging the power interruption. **Further information:** "Tool management", Page 139
RESET COLUMN T	Reset tool number T column Depends on optional machine parameter **showRe-setColumnT** (no. 125303)
BEGIN LINE ←	Go to beginning of line
END LINE →	Go to end of line
SELECT	Select a tool from the tool table: The control shows the contents of the tool table. Use the arrow keys to select a tool, press **OK** to transfer it to the pocket table
RESET COLUMN	Reset the value
EDIT CURRENT FIELD	Place the cursor in the current cell
SORT	Sort the view

⚙️ Refer to your machine manual.
The machine manufacturer defines the features, properties and designations of the various display filters.

Tool change

Automatic tool change

⚙️ Refer to your machine manual.

The tool change function can vary depending on the individual machine tool.

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the control reaches a tool call with **TOOL CALL**, it replaces the inserted tool by another from the tool magazine.

Overtime for tool life

⚙️ This function must be enabled and adapted by the machine manufacturer.

The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

The machine manufacturer specifies whether this column is enabled and how it is used during tool search.

Tool usage test

Requirements

⚙ Refer to your machine manual.
The tool-usage-test function must be enabled by your
machine manufacturer.

To conduct a tool usage test, you must activate **Create tool usage
files** in the MOD menu.

Further information: "Generating a tool usage file", Page 278

Generating a tool usage file

Depending on the setting in the MOD menu, you have the following options for generating the tool usage file:

- Simulate the complete NC program in the **Test Run** operating mode
- Completely run the NC program in the **Program Run, Full Sequence/Single Block** operating modes
- In the **Test Run** operating mode, press the **GENERATE TOOL USAGE FILE** soft key (also possible without simulation)

The tool usage file generated is in the same directory as the NC program. It contains the following information:

Column	Meaning
TOKEN	■ **TOOL**: Tool usage time per tool call. The entries are listed in chronological order.
	■ **TTOTAL**: Total usage time of a tool
	■ **STOTAL**: Call of a subprogram. The entries are listed in chronological order.
	■ **TIMETOTAL**: The total machining time of the NC program is entered in the **WTIME** column. In the **PATH** column the control saves the path name of the corresponding NC program. The **TIME** column shows the sum of all **TIME** entries (feed time without rapid traverse movements). The control sets all other columns to 0
	■ **TOOLFILE**: In the **PATH** column, the control saves the path name of the tool table with which you conducted the test run. This enables the control during the actual tool usage test to detect whether you performed the test run with TOOL.T
TNR	Tool number (**-1**: Tool not inserted yet)
IDX	Tool index
NAME	Tool name from the tool table
TIME	Tool usage time in seconds (feed time without rapid traverse movements)
WTIME	Tool-usage time in seconds (total usage time between tool changes)
RAD	**Tool radius R** + **Oversize of tool radius DR** from the tool table. (in mm)
BLOCK	Block number in which the **TOOL CALL** block was programmed
PATH	■ **TOKEN** = **TOOL**: Path name of the active main program or subprogram
	■ **TOKEN** = **STOTAL**: Path name of the subprogram
T	Tool number with tool index

Column	Meaning
OVRMAX	Maximum feed rate override that occurred during machining. The control enters the value 100 (%) during the test run
OVRMIN	Minimum feed rate override that occurred during machining. The control enters the value -1 during the test run
NAMEPROG	0: The tool number is programmed 1: The tool name is programmed

The control saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **dependentFiles** (no. 122101) is set to **MANUAL**

Using a tool usage test

Before starting a program in the **Program Run, Full Sequence/ Single Block** operating modes, you can check whether the tools being used in the selected NC program are available and have sufficient remaining tool life. The control then compares the actual tool-life values from the tool table with the nominal values from the tool usage file.

TOOL USAGE	▶ Press the **TOOL USAGE** soft key
TOOL USAGE TEST	▶ Press the **TOOL USAGE TEST** soft key > The control opens the **Tool usage test** pop-up window indicating the result of the usage test.
OK	▶ Press the **OK** soft key > The control closes the pop-up window.
ENT	▶ Alternative: Press the **ENT** key

You can query the tool usage test with the **FN 18 ID975 NR1** function.

4.2 Touch probe table

Application

The touch probe table **tchprobe.tp** defines the touch probe and data for the touching process, e.g. the probe feed rate. If you use several touch probes, you can save separate data for each touch probe.

Description of function

The touch probe table contains the parameters below:

Parameter	Meaning	Input
NO	Sequential number of touch probe This number assigns the touch probe in tool table column **TP_NO** to the data.	**1…99**
TYPE	**Selection of the touch probe?** ℹ️ 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.	**TS120**, **TS220**, **TS249**, **TS260**, **TS440**, **TS444**, **TS460**, **TS630**, **TS632**, **TS640**, **TS642-3**, **TS642-6**, **TS649**, **TS740**, **KT130**, **OEM**
CAL_OF1	**TS center misalignmt. ref. axis? [mm]**	**-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	**-99999.9999…+99999.9999**
CAL_ANG	**Spindle angle for calibration?**	**0.0000…+359.9999**
F	**Probing feed rate? [mm/min]** **F** can never be greater than defined in the optional machine parameter **maxTouchFeed** (no. 122602).	**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	**+10…+99999**
DIST	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will issue an error message.	**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.	**0.00100…+99999.99999**
F_PREPOS	**Pre-position at rapid? ENT/NOENT** Speed for pre-positioning: ▪ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ▪ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**FMAX_PROBE**, **FMAX_MACHINE**

Parameter	Meaning	Input
TRACK	**Probe oriented? Yes=ENT/No=NOENT**	**ON**, **OFF**
	■ **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.	
SERIAL	**Serial number?**	**Text width 15**
	The control automatically edits this parameter of touch probes with an EnDat interface.	
REACTION	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT**	**NCSTOP**, **EMERGSTOP**
	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	
STYLUS	**Shape of the stylus**	**SIMPLE**, **L-TYPE**

Editing the touch probe table

To edit the touch probe table:

▶ Press the **Manual operation** key

▶ Press the **TOUCH PROBE** soft key

> The control displays a soft key row for edge finder functions.

▶ Press the **TCH PROBE TABLE** soft key

> The control opens the touch probe table.
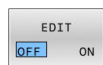
▶ Set the **EDIT** soft key to **ON**

▶ Select the desired value

▶ Perform desired changes
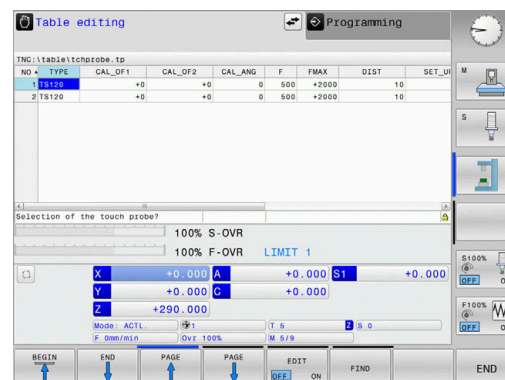
> **i** You can also edit the touch probe table values in the tool management.

4.3 Tool management

Basics

⚙️ Refer to your machine manual.

Tool management is a machine-dependent function, which may be partly or completely deactivated. The machine manufacturer defines the exact range of functions.

In tool management, your machine manufacturer can provide a wide range of functions for tool handling. Examples:

- Display and editing of all tool data from the tool table and the touch probe table
- Easily readable and adaptable representation of the tool data in fillable forms
- Any description of the individual tool data in the new table view
- Mixed representation of data from the tool table and the pocket table
- Fast sorting of all tool data by mouse
- Use of graphic aids, e.g. color coding of tool or magazine status
- Copying and pasting of all tool data pertaining to a tool
- Graphic depiction of tool type in the table view and in the detail view for a better overview of the available tool types

ℹ️
- If you edit a tool in tool management, the selected tool is locked. If this tool is required in the NC program being used, the control shows the message: **Tool table locked**.
- When using extended tool management, you can reset the pocket table only before acknowledging the power interruption.

  Further information: "Pocket table for tool changer", Page 129

Calling tool management

⚙ Refer to your machine manual.

The procedure for calling the tool management may differ from that described below.

▶ Select the tool table: Press the **TOOL TABLE** soft key

▶ Scroll through the soft-key row

▶ Press the **TOOL MANAGEMENT** soft key

> The control switches to the new table view.

Tool management view

In the new view, the control presents all tool information in the following four tabs:

- **Tools**: Tool-specific information
- **pockets**: Pocket-specific information

Editing tool management

The tool management can be operated by mouse or with the keys and soft keys:

Soft key	Editing functions for tool management
BEGIN ↑	Select the table start
END ↓	Select the table end
PAGE ↑	Select the previous page in the table
PAGE ↓	Select the next page in the table
FORM FOR TOOL	Call the form view of the marked tool. Alternative function: Press the **ENT** key
(tab icon)	Changing tab: **Tools** and **pockets**
FIND	Search function: Here you can select the column to be searched and then the search term either from a list or by entering it
TOOL IMPORT	Import tools
EXPORT TOOL	Export tools
DELETE MARKED TOOLS	Delete marked tools
APPEND N LINES AT END	Add several lines at end of table
UPDATE THE VIEW	Update table view
PROG. TOOL DISPLAY HIDE	Show the programmed tools column (if the **Pockets** tab is active)
COLUMN SORT MOVE	Define the settings: ■ **SORT COLUMN** active: Click the column header to sort the content of the column ■ **SHIFT COLUMN** active: The column can be moved by drag and drop
RESET SETTINGS	Reset the manually changed settings (move columns) to the original condition
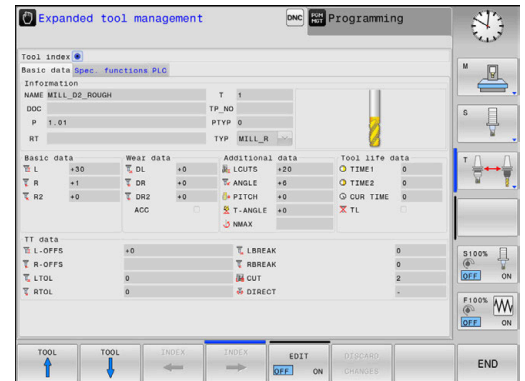
ⓘ   You can edit the tool data only in the form view. To activate the form view, press the **FORM FOR TOOL** soft key or the **ENT** key for the currently highlighted tool.

If you use the tool management without a mouse, then you can activate and deactivate functions with the **-/+** key.

In the tool management, use the **GOTO** key to search for the tool number or pocket number.

In addition, you can perform the following functions by mouse:

- Sorting function: You can sort the data in ascending or descending order (depending on the active setting of the soft key) by clicking a column of the table head.
- Arrange columns. You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down. The control does not save the current column sequence when you exit the tool management (depending on the active setting of the soft key).
- Show miscellaneous information in the form view: The control displays tool tips when you leave the mouse pointer on an active entry field for more than a second and when you have set the **EDIT ON/OFF** soft key to **ON**

Editing with active form view

If the form view is active, the following functions are available to you:

Soft key	Editing functions, form view
TOOL ↑	Select the tool data of the previous tool
TOOL ↓	Select the tool data of the next tool
INDEX ←	Select previous tool index (only active if indexing is enabled)
INDEX →	Select the next tool index (only active if indexing is enabled)
SELECT	Open a pop-up window for selection (only available for selection fields)
DISCARD CHANGES	Discard all changes made since the form was called
INSERT INDEX	Add tool index
DELETE INDEX	Delete tool index
COPY DATA RECORD	Copy the tool data of the selected tool
INSERT DATA REC.	Insert the copied tool data in the selected tool

Deleting marked tool data

Using this function you can simply delete tool data that you no longer need.

Follow the steps outlined below for deleting:

▶ In the tool management you use the arrow keys or mouse to mark the tool data that you wish to delete

▶ Press the **DELETE MARKED TOOLS** soft key

> The control shows a pop-up window listing the tool data to be deleted.

▶ Press the **EXECUTE** soft key to start the deletion procedure

> The control shows a pop-up window with the deletion status.

▶ Terminate the delete process by pressing the **END** key or soft key

---

NOTICE

Caution: Data may be lost!

The **DELETE MARKED TOOLS** function permanently deletes the tool data. The data is not automatically backed up by the control (e.g., to a recycle bin) before being deleted. The data is irreversibly deleted by this function.

▶ Regularly back up important data to external drives

---

i ■ The tool data of tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.

■ Make sure to keep the tool table as short and clear as possible so that it does not impair the computing speed of your control. Use a maximum of 10,000 tool entries in tool management. For example, you can delete all unused tool numbers; tool numbers need not be sequential.

Available tool types

i In tool management, the control only displays the input fields related to the selected tool type.

The tool management displays the various tool types with an icon. The following tool types are available:

Icon	Tool type	Tool type number
	Undefined,****	99
	Milling cutter,MILL	0
	Roughing cutter,MILL_R	9
	Finishing cutter,MILL_F	10

Icon	Tool type	Tool type number
	Face milling cutter,MILL_FACE	14
	Ball-nose cutter, BALL	22
	Toroid cutter, TORUS	23
	Chamfer mill, MILL_CHAMFER	24
	Side milling cutter, MILL_SIDE	25
	Drill,DRILL	1
	Tap,TAP	2
	Center drill,CENT	4
	Touch probe,TCHP	21
	Ream,REAM	3
	Countersink,CSINK	5
	Piloted counterbore,TSINK	6
	Boring tool,BOR	7
	Back boring tool,BCKBOR	8
	Thread mill,GF	15
	Thread mill w/ countersink,GSF	16
	Thread mill w/ single thread,EP	17
	Thread mill w/ indxbl insert,WSP	18
	Thread milling drill,BGF	19
	Circular thread mill,ZBGF	20

Importing and exporting tool data

Importing tool data

⚙   Refer to your machine manual.

The machine manufacturer can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs.

Using this function you can simply import tool data that you have measured externally on a presetting device, for example. The file to be imported must have the CSV format (**c**omma **s**eparated **v**alue). **CSV** files are text files designed for the exchange of data with a simple structure. Accordingly, the import file must have the following structure:

- **Line 1**: In the first line you define the column names in which the data defined in the subsequent lines is to be placed. The column names are separated with a comma.
- **Other lines**: All the other lines contain the data that you wish to import into the tool table. The order of the data must match the order of the column names in Line 1. The data is separated by commas, decimal numbers are to be defined with a decimal point.

Follow the steps outlined below for importing:

▶ Copy the tool table to be imported to the **TNC:\system\tooltab** directory on the hard disk of the control

▶ Start expanded tool management

▶ Press the **TOOL IMPORT** soft key in the tool management

> The control shows a pop-up window with the CSV files that are saved in the **TNC:\system\tooltab** directory

▶ Use the arrow keys or mouse to select the file to be imported and confirm with the **ENT** key

> The control shows a pop-up window with the content of the CSV file

▶ Start the import procedure with the **EXECUTE** soft key.

 ℹ️
- The CSV file to be imported must be stored in the **TNC: \system\tooltab** directory.
- If you import the tool data of existing tools (whose numbers are in the pocket table) the control issues an error message. You can then decide whether to skip this data record or insert a new tool. The control inserts a new tool into the first empty line of the tool table.
- If the imported CSV file contains unknown table columns, the control displays a message during import. An additional note informs you that the data will not be transferred.
- Make sure that the column designations have been specified correctly.
  Further information: "Entering tool data into the table", Page 124
- You can import any tool data, the associated data record does not have to contain all the columns (or data) of the tool table.
- The column names can be in any order, the data must be defined in the corresponding order.

Example

T,L,R,DL,DR	Line 1 with column names
4,125.995,7.995,0,0	Line 2 with tool data
9,25.06,12.01,0,0	Line 3 with tool data
28,196.981,35,0,0	Line 4 with tool data

Exporting tool data

Using this function you can simply export tool data to read it into the tool database of your CAM system, for example. The control stores the exported file in the CSV format (**c**omma **s**eparated **v**alue). **CSV** files are text files designed for the exchange of data with a simple structure. The export file has the following structure:

- **Line 1**: In the first line the control stores the column names of all the relevant tool data to be defined. The column names are separated from each other by commas.
- **Further lines**: All the other lines contain the data of the tools that you have exported. The order of the data matches the order of the column names in Line 1. The data is separated by commas, the control outputs decimal numbers with a decimal point.

Follow the steps outlined below for exporting:

- ▶ In the tool management you use the arrow keys or mouse to mark the tool data that you wish to export
- ▶ Press the **EXPORT TOOL** soft key
- > The control shows a pop-up window
- ▶ Enter a name for the CSV file and confirm it with the **ENT** key
- ▶ Start the export procedure with the **EXECUTE** soft key
- > The control shows a pop-up window with the status of the export process
- ▶ Terminate the export process by pressing the **END** key or soft key

> **i** By default the control stores the exported CSV file in the **TNC:\system\tooltab** directory.

4.4 Tool carrier management

Fundamentals

You can create and manage tool carriers using the tool carrier management. The control factors the tool carriers into the calculations.

> Refer to your machine manual.
>
> This function must be enabled and adapted by the machine manufacturer.
>
> The machine manufacturer defines the insertion point for the tool carrier.

On 3-axis machines, tool carriers of right-angled angle heads are useful for machining in the tool axes **X** and **Y**, as the control takes the dimensions of the angle heads into account.

You must carry out the following steps so that the control can factors the tool carriers into the calculations:

- Saving tool carrier templates
- Parameterizing tool carriers
- Assign the tool carrier

> If you are using M3D or STL files instead of parameterized tool carriers, the first two steps are not required.

> Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

Saving 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 so that you need not design all tool carriers yourself. Tool carrier templates are 3D models with fixed geometries but editable dimensions.

The tool carrier templates must be saved in **TNC:\system \Toolkinematics** and have the extension **.cft**.

Sample files for tool carrier templates are available on the programming station in the **TNC:\system\Toolkinematics** folder.

> 🛈 If the tool carrier templates are not available on your control, please download the data you require from:
> **HEIDENHAIN NC solutions**

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

> 🛈 The tool carrier templates may consist of several sub-files. If the sub-files are incomplete, the control will display an error message.
> **Do not use incomplete tool carrier templates!**

Parameterizing tool carriers

Before the control can factor the tool carrier into the calculations, you must give the tool carrier templates the actual dimensions. These parameters are entered in the additional **ToolHolderWizard** tool.

Save the parameterized tool carriers with the extension **.cfx** under **TNC:\system\Toolkinematics**.

The additional **ToolHolderWizard** tool is mainly operated with the mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics**, and **3D graphics** by holding down the left mouse button.

The following icons are available in the additional **ToolHolderWizard** tool:
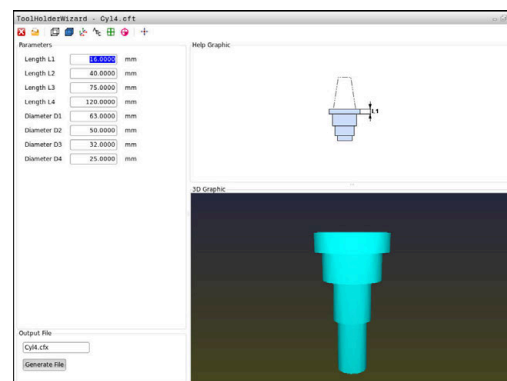
Icon	Function
✕	Close the additional tool
📁	Open file
⬛	Switch between wire frame model and solid object view
⬛	Switch between shaded and transparent view
⬚	Display or hide transformation vectors
ᴬ𝐵𝐶	Show or hide names of collision objects
⊞	Display or hide test points
⊕	Show or hide measurement points
⊹	Return to starting view of the 3D model

ℹ️ If the tool carrier template does not contain any transformation vectors, names, test points, or measurement points, then the additional **ToolHolderWizard** tool does not execute any function when the corresponding icons are activated.

Parameterizing tool carrier templates in Manual operation mode

To parameterize a tool carrier template and save these parameters:

[Manual operation icon]	▶ Press the **Manual operation** key
TOOL TABLE	▶ Press the **TOOL TABLE** soft key
EDIT OFF ON	▶ Press the **EDIT** soft key
→	▶ Move the cursor to the **KINEMATIC** column
SELECT	▶ Press the **SELECT** soft key
TOOL HOLDER WIZARD	▶ Press the **TOOL HOLDER WIZARD** soft key > The control opens the additional **ToolHolderWizard** tool in a pop-up window.
[Open file icon]	▶ Press the **OPEN FILE** icon > The control opens a pop-up window. ▶ Select the desired tool carrier template using the preview screen ▶ Press the **OK** button > The control opens the selected tool carrier template. > The cursor goes to the first parameterizable value. ▶ Adjust values ▶ Enter the name for the parameterized tool holder in the **Output file** area ▶ Press the **GENERATE FILE** button ▶ If required, reply to the message on the control
[Close icon]	▶ Press the **CLOSE** icon > The control closes the additional tool

Parameterizing tool carrier templates in Programming operating mode

Proceed as follows to parameterize tool carrier templates and save these parameters:

▶ Press the **Programming** key

▶ Press the **PGM MGT** key
▶ Select the path **TNC:\system\Toolkinematics**
▶ Select the tool carrier template
> The control opens the additional **ToolHolderWizard** tool with the selected tool carrier template.
> The cursor goes to the first parameterizable value.
▶ Adjust values
▶ Enter the name for the parameterized tool holder in the **Output file** area
▶ Press the **GENERATE FILE** button
▶ If required, reply to the message on the control
▶ Press the **CLOSE** icon
> The control closes the additional tool

Assigning a tool carrier

To allow the control to factor a tool carrier into calculations, you must assign the tool carrier to a tool and **call the tool again**.

> ℹ️ 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, error-free STL files or M3D files.**
>
> 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.
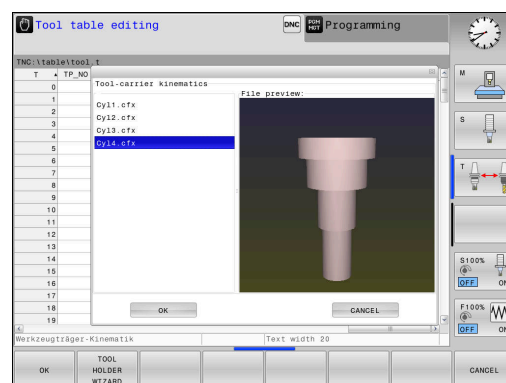
To assign a tool carrier to a tool:
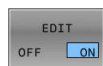
🖐	▶ Operating mode: Press the **Manual operation** key
TOOL TABLE	▶ Press the **TOOL TABLE** soft key
EDIT OFF ON	▶ Press the **EDIT** soft key
→	▶ Move the cursor to the **KINEMATIC** column of the required tool
SELECT	▶ Press the **SELECT** soft key > The control opens a pop-up window with the available tool carriers. ▶ Select the desired tool carrier using the preview ▶ Press the **OK** soft key > The control copies the name of the selected tool carrier to the **KINEMATIC** column.
END	▶ Exit the tool table

5

Setup

5.1   Switching on/off

Switch-on

> ⚠ **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.

Switch the machine and the control on as follows:

▶ Switch on the power supply for the control and the machine
> The control displays the switch-on status in the subsequent dialogs.
> If booting was successful, the control displays the **Power interrupted** dialog

CE   ▶ Press the **CE** key to clear the message
      > The control displays the **Compiling PLC program** dialog; the PLC program is compiled automatically
      > The control displays the **Switch on external dc voltage** dialog

I   ▶ Switch on the machine control voltage
      > The control carries out a self-test.

If the control does not register an error, it displays the **Traverse reference points** dialog.

If the control registers an error, it issues an error message.

Checking the axis position

> ⚙ This section applies only to machine axes with EnDat encoders.

If the actual axis position after the machine has been switched on does not match the shutdown position, the control displays a pop-up window.

▶ Check the axis position of the affected axis

▶ If the current axis position matches that proposed in the display, confirm with **YES**

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

---

Traverse reference points

If the control performs the self-test successfully, it then displays the **Traverse reference points** dialog.

> ⚙ Refer to your machine manual.
>
> Switching on the machine and traversing the reference points can vary depending on the machine tool.
>
> If your machine is equipped with absolute encoders, you can leave out crossing the reference points.

> ℹ If you intend only to edit or graphically simulate NC programs, you can select the **Programming** or **Test Run** operating mode immediately after switching on the control voltage, without needing to home the axes.
>
> You can neither set a preset nor modify a preset via the preset table without having homed the axes. The control issues the **Traverse reference points** message.
>
> You can traverse the reference points later. For this purpose, press the **PASS OVER REFERENCE** soft key in **Manual operation** mode.

Cross the reference points manually in the displayed sequence:

▶ For each axis press the **NC START** key, or
> The control is now ready for operation in the **Manual operation** operating mode.

As an alternative you can cross the reference points in any sequence:

▶ Press and hold the axis direction key for each axis until the reference point has been traversed
> The control is now ready for operation in the **Manual operation** operating mode.

Switching off

⚙ Refer to your machine manual.
Deactivation is a machine-dependent function.

To prevent data from being lost on switch-off, you need to shut down the operating system of the control as follows:

▶ Operating mode: Press the **Manual operation** key

▶ Press the **OFF** soft key

▶ Confirm with the **SHUT DOWN** soft key
▶ When the control displays the message **Now you can switch off** in a pop-up window, you may switch off the power supply to the control

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

5.2 Moving the machine axes

Note

⚙ Refer to your machine manual.
Movement of the axes via the axis direction keys can vary depending on the machine.

Moving the axis with the axis direction keys

🖐 ▶ Operating mode: Press the **Manual operation** key

X+ ▶ Press the axis-direction key and hold it down as long as you wish the axis to move

X+ ▶ Alternatively, move the axis continuously: Press and hold the axis-direction key and press the **NC Start** key

🔲

🔲 ▶ To stop: Press the **NC Stop** key

You can change the feed rate at which the axes are moved with the **F** soft key.

Further information: "Spindle speed S, feed rate F and miscellaneous function M", Page 172

If a moving task is active on the machine, the control displays the **control in operation** symbol.

Incremental jog positioning

Incremental jog positioning allows you to move a machine axis by a preset distance.

The input range for the infeed is from 0.001 mm to 10 mm.

▶ Operating mode: Press the **Manual operation** or **Electronic handwheel** key

▶ Shift the soft-key row

▶ Select incremental jog positioning: Switch the **INCREMENT** soft key to **ON**

▶ Enter the infeed of the **linear axes**

▶ Confirm with the **CONFIRM VALUE** soft key

▶ Or confirm with the **ENT** key

▶ Confirm with the **OK** soft key

> The jog increment is active.

> The control displays the set values in the upper area of the screen.
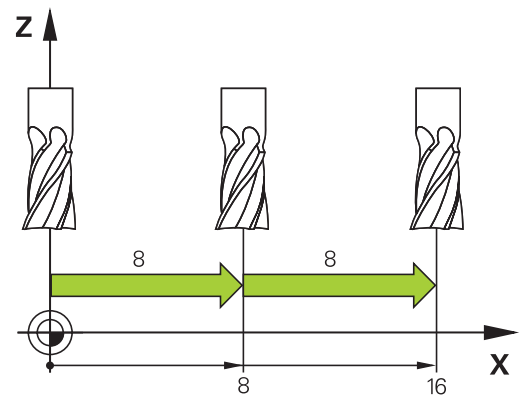
Canceling incremental jog positioning

▶ Set the **INCREMENT** soft key to **OFF**

ℹ️ If you are in the **Jog increment** menu, you can switch off incremental jog positioning with the **SWITCH OFF** soft key.

Traverse with the HR 510 electronic handwheel

The portable HR 510 handwheel is equipped with two permissive buttons. The permissive buttons are located above the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 510 handwheel features the following operating elements:

1 EMERGENCY STOP button

2 Handwheel

3 Permissive buttons

4 Axis address keys

5 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine manufacturer)

6 Direction in which the control moves the selected axis

7 Machine functions (set by the machine manufacturer)

Traversing axes

Red LEDs show the active functions, such as the selected axis

▶ Select the **Electronic handwheel** operating mode

▶ Press and hold a permissive button
▶ Select the axis

▶ Select the feed rate

▶ Move the active axis in the positive direction

▶ Move the active axis in the negative direction

Traverse with electronic handwheels with display

> **⚠ DANGER**
>
> **Caution: hazard to the user!**
>
> Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!
>
> ▶ Devices should be connected or removed only by authorized service technicians
> ▶ Only switch on the machine via a connected handwheel or a secured connection

The control supports traversing with the following electronic handwheels:

- HR 520: Handwheel with display, data transmission via cable
- HR 550FS: Handwheel with display, data transmission via radio

> ⚙ Your machine manufacturer can make additional functions of the HR 5xx handwheels available.

The HR 520 and HR 550FS portable handwheels feature a display that allows the control to show different types of information. In addition, you can use the handwheel soft keys for important setup functions (e.g., setting of presets or entering and running M functions).

As soon as you have activated the handwheel with the handwheel activation key, the operating panel is locked. The control shows this status in a pop-up window on the screen.

1 EMERGENCY STOP key

2 Handwheel display for status and for selecting functions

3 Soft keys

4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration

5 Permissive button

6 Arrow keys for defining handwheel sensitivity

7 Handwheel activation key

8 Key for traverse direction of the selected axis

9 Rapid traverse superimposing for the axis direction key

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 550 FS wireless handwheel
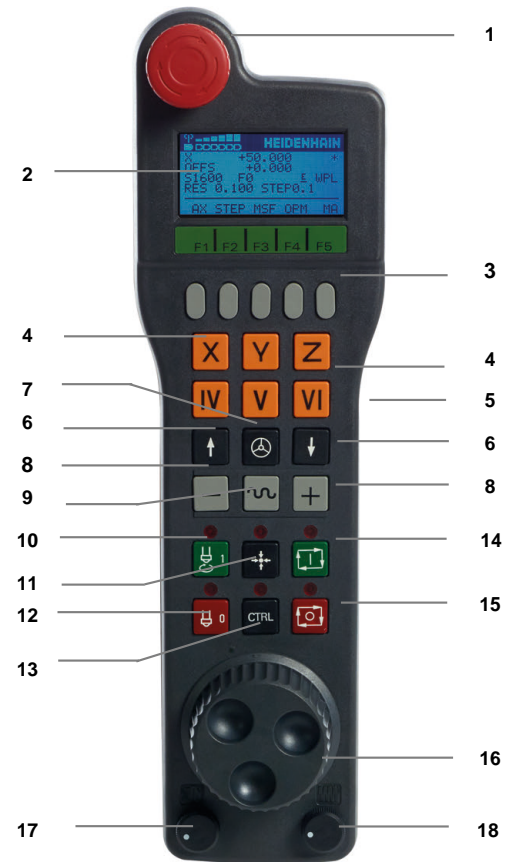
Handwheel display

1 **Only with the HR 550 FS wireless handwheel**: Shows whether the handwheel is in its docking station or whether radio mode is active

2 **Only with the HR 550 FS wireless handwheel**: Display of field strength; six bars = maximum field strength

3 **Only with the HR 550 FS wireless handwheel**: Charging state of battery; six bars = maximum charge. A bar moves from the left to the right during recharging

4 **ACTL**: Type of position display

5 **Y+129.9788**: Position of the selected axis

6 *: Control in operation; program run has been started or axis is in motion

7 **S0:**: Current spindle speed

8 **F0:** Feed rate at which the selected axis is moving

9 **E**: Error message

If an error message appears on the control, the handwheel display shows the **ERROR** message for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.

10 **RES 5.0**: Active handwheel resolution. Path traversed by the selected axis with one handwheel revolution

11 **STEP ON** or **OFF**: Incremental jog active or inactive. If the function is active, the control additionally displays the current traversing step

12 Soft-key row: Selection of various functions, described in the following sections
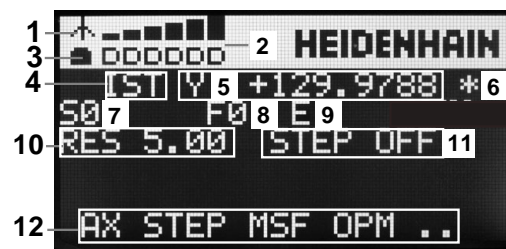
Special features of the HR 550 FS wireless handwheel

> # ⚠ 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)

> ℹ Due to various potential sources of interference, a wireless connection is not as reliable as a cable connection. Before a wireless handwheel can be used, it must be checked whether there is an overlapping with other wireless devices. If this is the case, then such overlapping must be eliminated. This inspection for the presence of radio frequencies or channels is obligatory for all industrial radio systems.
>
> When you do not use the HR 550 FS, always place it into the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.
>
> If an error (interruption of the radio connection, poor reception quality, defective handwheel component) occurs, the handwheel always reacts with an emergency stop.

The HR 550FS wireless handwheel features a rechargeable battery. The battery starts charging when you put the handwheel in the holder.

The HR 550FS can be operated by battery for up to eight hours before it needs recharging. When you aren't using the handwheel, we recommend that you place it in the handwheel holder.

As soon as the handwheel is in its holder, it switches internally to cable operation. This means you can still use it even if the handwheel is fully discharged. The functions are the same as with wireless operation.

> A completely discharged handwheel takes approx. three hours for a full charge.
>
> Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.

The transmission range is amply dimensioned. If you happen to get close to the limit of the transmission range, for example with very large machines, the HR 550FS will set off a vibrating alarm that is reliably noticed. If this happens you must reduce the distance to the handwheel holder in which the radio receiver is integrated.

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

If the control has triggered an emergency stop you must reactivate the handwheel. Proceed as follows:

MOD

▶ Press the **MOD** key

> The control opens the MOD menu.

▶ Select the **Machine settings** group

SET UP WIRELESS HANDWHEEL

▶ Press the **SET UP WIRELESS HANDWHEEL** soft key

> The control opens a pop-up window for configuring wireless handwheels.

▶ Press the **Start handwheel** button to reactivate the wireless handwheel

▶ Select the **END** button

The MOD menu includes a function for initial operation and configuration of the handwheel.

Further information: "Configuring the HR 550\FS radio handwheel", Page 281

Selecting the axis to be moved

You can directly use the axis keys to activate the main axes X, Y, Z, and three other axes that can be defined by the machine manufacturer. Your machine manufacturer can also place the virtual axis VT directly on one of the free axis keys. If the virtual axis VT is not on one of the axis keys, proceed as follows:

▶ Press the handwheel soft key **F1** (**AX**)

> The control shows all active axes on the handwheel display. The currently active axis flashes.

▶ Select the desired axis with the **F1** (**->**) or **F2** soft keys (**<-**) and confirm with the **F3** (**OK**) handwheel soft key

Setting the handwheel sensitivity

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).

Taking the example of a defined handwheel speed of 1, the following handwheel sensitivity values result from the speed levels for the respective units of measure:

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 inches/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 inches/revolution

Moving the axes

▶ Activate the handwheel: Press the handwheel key on the HR 5xx

> You can now operate the control only via the HR 5xx. The control displays a pop-up window with this information on the screen

▶ Select the desired operating mode with the **OPM** soft key as needed

▶ Press and hold the permissive button as needed

▶ Use the handwheel to select the axis to be moved. Select the additional axes with the soft keys as needed

▶ Move the active axis in the positive direction, or

▶ Move the active axis in the negative direction

▶ Deactivate the handwheel: Press the handwheel key on the HR 5xx

> Now you can operate the control via the operating panel again.

Potentiometer settings

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 between the potentiometers, the feed rate is either frozen or reduced.

If the feed rate before switching is higher than the feed rate after switching, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching is less than the feed rate after switching, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because the activated feed-rate potentiometer will only then be effective.

Incremental jog positioning

With incremental jog positioning the control moves the currently active handwheel axis by a preset increment defined by you:

▶ Press the handwheel soft key F2 (**STEP**)

▶ Activate incremental jog positioning: Press the handwheel soft key 3 (**ON**)

▶ Select the desired jog increment by pressing the **F1** or **F2** key. 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 soft key 4 (**OK**)

▶ Use the **+** or **-** handwheel key to move the active handwheel axis in the corresponding direction

i 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**.

Entering miscellaneous functions M

▶ Press the handwheel soft key **F3** (**MSF**)

▶ Press the handwheel soft key **F1** (**M**)

▶ Select the desired M function number by pressing the **F1** or **F2** key

▶ Execute the M miscellaneous function by pressing the **NC Start** key

Entering the spindle speed S

▶ Press the handwheel soft key **F3** (**MSF**)

▶ Press the handwheel soft key **F2** (**S**)

▶ Select the desired spindle speed by pressing the **F1** or **F2** key

▶ Activate the new spindle speed S with the **NC Start** key

i 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**.

Entering the feed rate F

▶ Press the handwheel soft key **F3** (**MSF**)

▶ Press the 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 handwheel soft key **F3** (**OK**)

> 🛈 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**.

Setting a preset

▶ Press the handwheel soft key **F3** (**MSF**)

▶ Press the handwheel soft key **F4** (**PRS**)

▶ If necessary, select the axis in which the preset will be set

▶ Zero the axis with the handwheel soft key **F3** (**OK**) or set the desired value with the handwheel soft keys **F1** and **F2**, and then confirm with the handwheel soft key **F3** (**OK**). By additionally pressing the **CTRL** key, you can increase the counting increment to 10

Switching between operating modes

The handwheel soft key **F4** (**OPM**) allows you to use the handwheel to switch the operating mode if the current status of the control permits a change in operating mode.

▶ Press the handwheel soft key **F4** (**OPM**)

▶ Select the desired operating mode by handwheel soft key

■ MAN: **Manual operation**
   MDI: **Positioning w/ Manual Data Input**
   SGL: **Program run, single block**
   RUN: **Program run, full sequence**

Generating a complete traversing block

> ⚙ Your machine manufacturer can assign any function to the **Generate NC block** handwheel key.

▶ Select the **Positioning w/ Manual Data Input** operating mode

▶ If necessary, use the arrow keys on the alphabetic keyboard to select the NC block after which the new positioning block should be inserted

▶ Activate the handwheel

▶ Press the **Generate NC block** key on the handwheel

> The control inserts a complete traversing block containing all axis positions selected through the MOD function.

Functions in the Program Run operating modes

You can use the following functions in Program Run operating modes:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After you have pressed the **NC Stop** key: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After you have pressed the **NC Stop** key: Manual traverse of axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during a program interruption (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys, which function similarly to the screen soft keys, are used for operating.

5.3 Spindle speed S, feed rate F and miscellaneous function M

Application

In the **Manual operation** and **Electronic handwheel** operating modes, you can enter the spindle speed S, feed rate F, and the M functions (miscellaneous functions) with soft keys.

Further information: "Enter a miscellaneous function M", Page 254

⚙ Refer to your machine manual.

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

Enter values

Spindle speed S, miscellaneous function M

To enter the spindle speed:

S	▶ Press the **S** soft key
	> The control shows the **Spindle speed S =** dialog in a pop-up window.
⟱	▶ Enter **1000** (spindle speed)
	▶ Press the **NC start** key to load the value

To start the spindle rotation with the specified speed **S**, use an **M** function. The **M** function value can be entered in the same way.

The control shows the current spindle speed in the status display. If the spindle speed is less than 1000, the control also shows a decimal place that has been entered.

Feed rate F

To enter the feed rate:

F	▶ Press the **F** soft key
	> The control opens a pop-up window.
	▶ Enter the desired feed rate
ENT	▶ Confirm with the **ENT** key

The following is valid for feed rate F:

- If you enter F=0, then the feed rate that the machine manufacturer has defined as minimum feed rate is active
- If the feed rate you entered exceeds the maximum value that has been defined by the machine manufacturer, then the value defined by the machine manufacturer is active
- F is retained even after a power interruption
- The control displays the contouring feed rate as follows:

The control shows the current feed rate in the status display.

- If the feed rate is less than 10, the control also shows one decimal place (if one was entered).
- The control displays two decimal places if the feed rate is less than 1.

Changing the spindle speed and feed rate

With the potentiometers you can vary the spindle speed **S** and feed rate **F** from 0% to 150% of the set value.

The feed-rate potentiometer only reduces the programmed feed rate and not the feed rate calculated by the control.

⚙ The override for spindle speed is only functional on machines with infinitely variable spindle drive.

Feed-rate limit F MAX

⚙ Refer to your machine manual.
The feed-rate limit depends on the machine.

The **F MAX** soft key enables you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you enter remains active after switch-off or switch-on.

If a feed-rate limit is active, the control displays an exclamation mark behind the feed rate-value in the status display.

Further information: "General status display", Page 68

The **F MAX** soft key is available in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Positioning w/ Manual Data Input**

Procedure

To activate the feed-rate limit F MAX:

▶ Operating mode: Press the **Positioning w/ Manual Data Input** key

▶ Set the **F MAX** soft key to **ON**
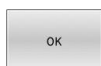
▶ Enter the desired maximum feed rate
▶ Press the OK soft key
> The control displays an exclamation mark behind the feed rate-value in the status display.

5.4 Preset management

Note

It is essential that you use the preset table in the following cases:

- If up to now you have been working with older controls with REF-based datum tables
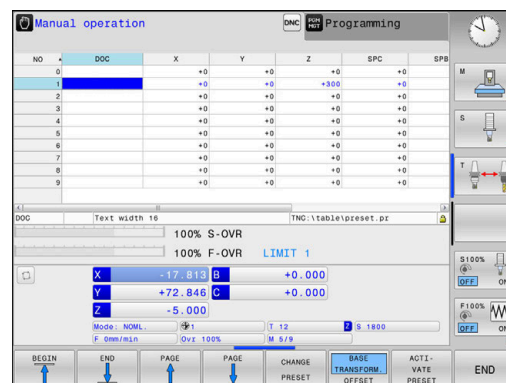
The preset table can contain any number of rows (presets). To optimize the file size and the processing speed, only use as many rows as you need to manage your presets.

For safety reasons, new rows can be inserted only at the end of the preset table.

Refer to your machine manual.

The machine manufacturer specifies the default values to be used for each column of a new line.

Creating and activating an INCH-based preset table

ⓘ   If you choose to switch the units for your control to **INCH**, the unit of measure for the preset table will not be adjusted automatically.

   If you want to change the unit of measure in the table as well, you need to create a new preset table.

To create and activate an **INCH**-based preset table:

[⇨]
▶ Select the **Programming** operating mode

[PGM MGT]
▶ Open the file manager
▶ Open the **TNC:\table** folder
▶ Rename the **preset.pr** file, e.g. to **preset_mm.pr**
▶ Create a new file and name it **preset_inch.pr**

[INCH]
▶ Select the **INCH** unit of measure
> The control opens the new, empty preset table.
> The control displays an error message related to a missing prototype file.
▶ Delete the error message

[APPEND N LINES AT END]
▶ Append lines, e.g. 10 lines
> The control adds the lines
▶ Position the cursor in the **ACTNO** column of line **0**
▶ Enter **1**

[ENT]
▶ Confirm your input

[PGM MGT]
▶ Open the file manager
▶ Rename the **preset_inch.pr** file to **preset.pr**

[✋]
▶ Select the **Manual operation** mode

[PRESET MANAGEMENT ✛]
▶ Open Preset management
▶ Review the preset table

ⓘ   The tool table is another table where the unit of measure is not adjusted automatically.

   Further information: "Creating and activating an INCH-based tool table", Page 123

Saving presets in the table

> ⚙ Refer to your machine manual.
>
> The machine manufacturer can disable presetting in individual axes.
>
> The machine manufacturer can define a different path for the preset table.
>
> In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.

The preset table has the name **PRESET.PR**, and is saved in the **TNC: \table** directory by default.

PRESET.PR is editable in the **Manual operation** and **Electronic handwheel** modes only if the **CHANGE PRESET** soft key was pressed. You can open the **PRESET.PR** preset table in the **Programming** operating mode but not edit it.

There are several methods for saving presets in the preset table:

- Manual input
- Using the touch probe cycles in the **Manual operation** and **Electronic handwheel** operating modes

> ℹ Operating notes:
> - In row 0 the control always saves the preset that you most recently set manually via the axis keys or via soft key. If the preset set manually is active, the control displays the text **PR MAN(0)** in the status display.

Copying the preset table

Copying the preset table into another directory (for data backup) is permitted. Write-protected lines are also write-protected in the copied tables.

Do not change the number of lines in the copied tables! If you want to reactivate the table, this may lead to problems.

To activate the preset table copied to another directory you have to copy it back.

If you select a new preset table, you need to reactivate the preset.

Manually saving the presets in the preset table

Proceed as follows in order to save presets in the preset table:

▶ Select the **Manual operation** mode

[X+]
[Y+]
[Z-]

▶ Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly

| PRESET MANAGEMENT ⊕ |

▶ Press the **PRESET MANAGEMENT** soft key

> The control opens the preset table and positions the cursor in the line of the active preset.

| CHANGE PRESET |

▶ Press the **CHANGE PRESET** soft key

> The control displays all available input options in the soft-key row.

| ↓ |

▶ Select the line in the preset table that you want to change (the line number is the preset number)

| → |

▶ If needed, select the column in the preset table that you want to change

| CORRECT THE PRESET |

▶ Use the soft keys to select one of the available input options

Input options

Soft key	Function
	Directly transfer the actual position of the tool (the measuring dial) as the new preset: This function only saves the preset in the axis in which the cursor is currently positioned
ENTER PRESET AGAIN	Assign any value to the actual position of the tool (the measuring dial): This function only saves the preset in the axis in which the cursor is currently positioned. Enter the desired value in the pop-up window
CORRECT THE PRESET	Incrementally shift a preset already stored in the table: This function only saves the preset in the axis in which the cursor is currently positioned. Enter the desired compensation value with the correct sign in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered value to mm
EDIT CURRENT FIELD	Directly enter the new preset without calculation of the kinematics (axis-specific). This function only saves the value in the axis in which the cursor is currently positioned. Enter the desired value in the pop-up window. If inch display is active: Enter the value in inches, and the control will internally convert the entered value to mm
SAVE ACTIVE PRESET	Write the currently active preset to a selectable line in the table: This function saves the preset in all axes, and then activates the appropriate line in the table automatically. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm

Editing the preset table

Soft key	Editing function in table mode
BEGIN	Select the table start
END	Select the table end
PAGE	Select the previous page in the table
PAGE	Select the next page in the table
CHANGE PRESET	Select the functions for entry of presets
ACTI- VATE PRESET	Activate the preset of the selected line of the preset table
APPEND N LINES AT END	Add several lines at end of table
COPY FIELD	Copy the field that is currently highlighted
PASTE FIELD	Insert the copied field
RESET LINE	Reset the selected line: The control enters **-** into all columns.
INSERT LINE	Insert a line as last line in the table
DELETE LINE	Delete the last line in the table

Protecting presets from being overwritten

You can use the **LOCKED** column to protect any lines in the preset table from being overwritten. The write-protected lines are color-highlighted in the preset table.

If you want to overwrite a write-protected line with a manual touch probe cycle, confirm with **OK** and enter the password (where password-protected).

> ### *NOTICE*
>
> **Caution: Data may be lost!**
>
> Lines that were locked with the **LOCK / UNLOCK PASSWORD** function can be unlocked only with the selected password. Forgotten passwords cannot be reset. This means that locked lines would be locked permanently. The preset table would thus no longer be fully usable.
>
> ▶ Prefer the alternative function **LOCK / UNLOCK**
> ▶ Note down your passwords

Proceed as follows to protect a preset from being overwritten:

CHANGE PRESET   ▶ Press the **CHANGE PRESET** soft key

→   ▶ Select the **LOCKED** column

EDIT CURRENT FIELD   ▶ Press the **EDIT CURRENT FIELD** soft key

Protection for a preset without using a password:

LOCK / UNLOCK   ▶ Press the **LOCK / UNLOCK** soft key
> The control writes an **L** in the **LOCKED** column.

Use a password to protect a preset:

LOCK / UNLOCK PASSWORD   ▶ Press the **LOCK / UNLOCK PASSWORD** soft key

▶ Enter the password in the pop-up window
OK   ▶ Confirm with the **OK** soft key or with the **ENT** key:
> The control writes **###** in the **LOCKED** column.

Canceling write-protection

To edit a line you have previously write-protected:

| CHANGE PRESET | ▶ Press the **CHANGE PRESET** soft key |

| → | ▶ Select the **LOCKED** column |

| EDIT CURRENT FIELD | ▶ Press the **EDIT CURRENT FIELD** soft key |

Preset protected without a password:

| LOCK / UNLOCK | ▶ Press the **LOCK / UNLOCK** soft key |
| | > The control cancels the write-protection. |

Preset protected with a password:
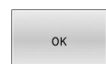
| LOCK / UNLOCK PASSWORD | ▶ Press the **LOCK / UNLOCK PASSWORD** soft key |

▶ Enter the password in the pop-up window

| OK | ▶ Confirm with the **OK** soft key or with the **ENT** key |
| | > The control cancels the write-protection. |

Activating a preset

Activating a preset in the Manual operation mode

NOTICE

Caution: Significant property damage!

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

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

▶ For undefined columns, enter values (e.g., **0**)

▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

---

ℹ Operating notes:
- When activating a preset from the preset table, the control resets any active datum shift, mirroring, or scaling factor.
- If you edit the value of the **DOC** column, then the preset must be reactivated. Only then does the control apply the new value.

▶ Select the **Manual operation** mode

▶ Press the **PRESET MANAGEMENT** soft key

▶ Select the preset number that you want to activate

▶ Or, with the **GOTO** key, select the preset number that you want to activate

▶ Confirm with the **ENT** key

▶ Press the **ACTI- VATE PRESET** soft key

▶ Confirm activation of the preset
> The control sets the display.

▶ Exit the preset table

Activating a preset in an NC program

Use Cycle **247** or the **PRESET SELECT** function in order to activate presets from the preset table during a program run.

In Cycle **247**, you can specify the number of the preset to be activated. With the **PRESET SELECT** function, you can specify the number of the preset or the entry in the **Doc** column that you want to activate.

Further information: Conversational Programming User's Manual

5.5 Presetting without a 3D touch probe

Note

When presetting, you set the control display to the coordinates of a known workpiece position.

> **i** All manual probe functions are available with a 3D touch probe.
>
> **Further information:** "Presetting with a 3D touch probe (option number 17)", Page 202

> **⚙** Refer to your machine manual.
>
> The machine manufacturer can disable presetting in individual axes.

Preparation

▶ Clamp and align the workpiece
▶ Insert the zero tool with known radius into the spindle
▶ Ensure that the control is showing the actual positions

Presetting with an end mill

▶ Select **Manual operation** mode

▶ Move the tool slowly until it touches (scratches) the workpiece surface

Setting a preset in an axis:

▶ Select the axis

> The control opens the **PRESETTING Z=** dialog window

▶ Alternative: Press the **SET PRESET** soft key

▶ Select the axis via soft key

▶ Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness d of the shim. In the tool axis: Consider the tool radius
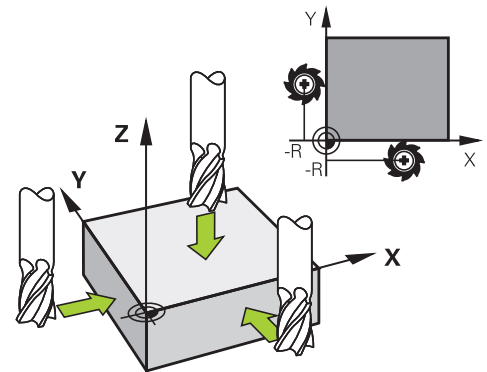
Repeat the process for the remaining axes.

If the tool in the tool axis has already been set, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

ⓘ Operating notes:
- The control automatically saves the preset set with the axis keys in row 0 of the preset table.
- If the machine manufacturer has locked an axis, then you cannot set a preset in that axis. The soft key for that axis is then not visible.

Using touch probe functions with mechanical probes or measuring dials

If you do not have an electronic 3D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

Further information: "Using a 3D touch probe (option 17)", Page 187

In place of the electronic signal generated automatically by a 3D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key.

Proceed as follows:
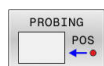
▶ Select any touch probe function by soft key

▶ Move the mechanical probe to the first position to be captured by the control.

▶ To capture the position: Press the **Actual-position-capture** soft key

> The control saves the current position.

▶ Move the mechanical probe to the next position to be captured by the control.

▶ To capture the position: Press the **Actual-position-capture** soft key

> The control saves the current position.

▶ If required, move to additional positions and capture as described previously

▶ **Preset**: In the menu window, enter the coordinates of the new preset, confirm with the **SET PRESET** soft key, or write the values to a table
  Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
  Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

▶ Terminate the probing function: Press the **END** key

ⓘ 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.

5.6 Using a 3D touch probe (option 17)

Overview

⚙️ Refer to your machine manual.

The control must be specifically prepared by the machine manufacturer for the use of a 3D touch probe.

The manual touch probe functions are available only with software option 17 (Touch Probe Functions).

If you are using a HEIDENHAIN touch probe with EnDat interface, then the software option Touch Probe Functions (option 17) is automatically enabled.

ℹ️ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

The following touch probe cycles are available in the **Manual operation** mode:

Soft key	Function	Page
CALIBRATE TS	Calibrating the 3-D Touch Probe	195
PROBING POS	Setting the preset on any axis	203
PROBING CC	Setting a circle center as preset	204
PROBING CL	Setting the centerline as preset	207
TCH PROBE TABLE	Touch probe system data management	137

Traverse movements with a handwheel with display

With a handwheel with display, it is possible to transfer control to the handwheel during a manual touch probe cycle.

Proceed as follows:

▶ Start the manual touch probe cycle
▶ Position the touch probe at a position near the first touch point
▶ Probe the first touch point
▶ Activate the handwheel on the handwheel
> The control shows the **Handwheel active** pop-up window.
▶ Position the touch probe at a position near the second touch point
▶ Deactivate the handwheel on the handwheel
> The control closes the pop-up window.
▶ Probe the second touch point
▶ If necessary, set the preset
▶ End the probing function

> ℹ️   If the handwheel is active, you cannot start the touch probe cycles.

Suppressing touch probe monitoring

Suppressing touch probe monitoring

If the control does not receive a stable signal from the touch probe, the **TCH PROBE MONITOR OFF** soft key is displayed.

Proceed as follows to deactivate touch-probe monitoring:

| | ▶ Select **Manual operation** mode |

| TCH PROBE MONITOR OFF | ▶ Press the **TCH PROBE MONITOR OFF** soft key
> 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. |

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.

> **i** 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.

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 operation** operating mode

---

Functions in touch probe cycles

Soft keys that are used to select the probing direction or a probing routine are displayed in the manual touch probe cycles. The soft keys displayed vary depending on the respective cycle:

Soft key	Function
X+	Select the probing direction
	Capture the actual position
	Probe hole (inside circle) automatically
	Probe stud (outside circle) automatically
PROBING CC	Probe a model circle (center point of several elements)
	Select a paraxial probing direction for probing of holes, studs and model circles

Automatic probing routine for holes, studs, and circular patterns

NOTICE
Danger of collision!
The control does not perform an automatic collision check with the stylus. During automatic probing procedures the control positions the touch probe to the probing positions automatically. There is a risk of collision if pre-positioning was not correct or if obstacles have been ignored.
▶ Program a suitable pre-position
▶ Use safety clearances to take obstacles into account

If you use a probing routine for automatic probing of a hole, stud, or a circular pattern, the control opens a form with the required entry fields.

Input fields in the Measure stud and Measure hole forms

Input field	Function
Stud diameter? or **Hole diameter**	Diameter of probe contact (optional for holes)
Safety clearance?	Distance to the probe contact in the plane
Incr. clearance height?	Positioning of touch probe in spindle axis direction (starting from the current position)

Automatic probing routine:

▶ Pre-position touch probe
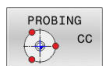
▶ Select the probing function: Press the **PROBING CC** soft key

▶ Hole should be probed automatically: Press the **HOLE** soft key
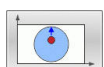
▶ Select paraxial probing direction

▶ Start probing function: Press the **NC Start** key
> The control carries out all pre-positioning and probing processes automatically.

The control approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.

i Operating and programming notes:

- Before starting an automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance opposite to the probing direction. The safety clearance is derived from the sum of the values in the touch-probe table and in the entry form.

- For inside circles with large diameters, the control can also position the touch probe on a circular arc at the feed rate **FMAX**. This requires that you enter a safety clearance for prepositioning and the hole diameter in the input form. Position the touch probe inside the hole at a position that is offset by approximately the safety clearance from the wall. Take the starting angle of the first probing process into account in pre-positioning; for example, at a starting angle of 0° the control will first probe in the positive direction of the reference axis.

- 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.

Selecting the probing cycle

▶ Select the **Manual operation** or **Electronic handwheel** mode of operation

| TOUCH PROBE | ▶ Select the probing functions: Press the **TOUCH PROBE** soft key |

| PROBING POS | ▶ Select the touch probe cycle by pressing the appropriate soft key, for example **PROBING POS** |
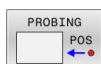
> The control displays the associated menu.

i Operating notes:

- When you select a manual probing function, the control opens a form displaying all data required. The content of the forms varies depending on the respective function.

- You can also enter values in some of the fields. Use the arrow keys to switch to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited are dimmed.

Recording measured values from the touch probe cycles

⚙️ Refer to your machine manual.
The control must be specially prepared by the machine manufacturer for use of this function.

After executing the respective touch-probe cycle, the control writes the measured values to 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:**.

ℹ️ Operating notes:
- If you run several touch probes cycles in a row, the control stores the measured values below each other.

Writing measured values from the touch probe cycles to a datum table

ℹ️ If you want to save measured values in the workpiece coordinate system, use the **ENTRY IN DATUM TABLE** function. If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function.

Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

With the **ENTRY IN DATUM TABLE** soft key, the control can write the values measured during any touch-probe cycle to a datum table:

▶ Select any probing function
▶ Enter the desired coordinates for the datum in the designated input boxes (depends on the touch probe cycle being run)
▶ Enter the datum number in the **Number in table?** input field
▶ Press the **ENTRY IN DATUM TABLE** soft key
> The control saves the datum in the indicated datum table under the entered number.

Writing measured values from the touch probe cycles to the preset table

ℹ️ If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function. If you want to save measured values in the workpiece coordinate system, use the **ENTRY IN DATUM TABLE** function.

Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193

With the **ENTRY IN PRESET TABLE** soft key, the control can write the values measured during any touch probe cycle to the preset table. The measured values are then stored referenced to the machine coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the TNC:\table\ directory.

▶ Select any probe function

▶ Enter the desired coordinates for the preset in the designated input boxes (depends on the touch probe cycle being run)

▶ Enter the preset number in the **Number in table?** input field

▶ Press the **ENTRY IN PRESET TABLE** soft key

> The control opens the **Overwrite active preset?** menu.

▶ Press the **OVERWRITE PRESET** soft key

> The control saves the preset in the preset table under the entered number.

- Preset number does not exist: The control does not save the line until the **CREATE LINE** soft key has been pressed (Create line in table?)

- Preset number is protected: Press the **ENTRY IN LOCKED LINE** soft key to overwrite the active preset

- Preset number is password-protected: Press the **ENTRY IN LOCKED LINE** soft key and enter the password to overwrite the active preset

ℹ️ The control displays a note if a table row cannot be written to because of disabling. The probing function itself is not interrupted.
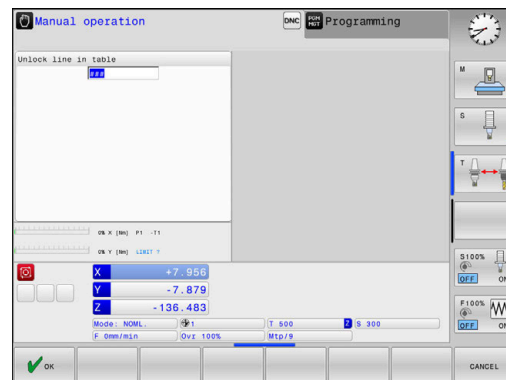
5.7 Calibrating 3D touch probes (option 17)

Introduction

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

> **i** Operating notes:
> - Always calibrate the touch probe again in the following cases:
>   - 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
> - When you press the **OK** soft key after calibration, the calibration values are applied to the active touch probe. The updated tool data then become immediately effective and it is not necessary to repeat the tool call.

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 control provides calibration cycles for calibrating the length and the radius:

TOUCH PROBE	▶ Press the **TOUCH PROBE** soft key
CALIBRATE TS	▶ Display the calibration cycles: Press **CALIBRATE TS**
	▶ Select the calibration cycle

Calibration cycles

Soft key	Function	Page
	Calibrating the length	196
	Measure the radius and the center offset using a calibration ring	197
	Measure the radius and the center offset using a stud or a calibration pin	197
	Measure the radius and the center offset using a calibration sphere	197

Calibrating the effective length

ℹ️ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

⚙️ The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.

▶ Set the preset in the spindle axis such that the following applies to the machine table: Z = 0
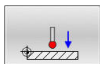
▶ Select the calibrating function for the touch probe length: Press the **TS calibration of tool length** soft key

> The control displays the current calibration data.

▶ **Datum for length?**: Enter the height of the ring gauge in the menu window

▶ Move the touch probe to a position just above the ring gauge

▶ To change the traverse direction (if necessary), press a soft key or an arrow key

▶ Probe surface: Press **NC Start** key

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **CANCEL** soft key to terminate the calibrating function.

> The control logs the calibration process in the TCHPRMAN.html file.
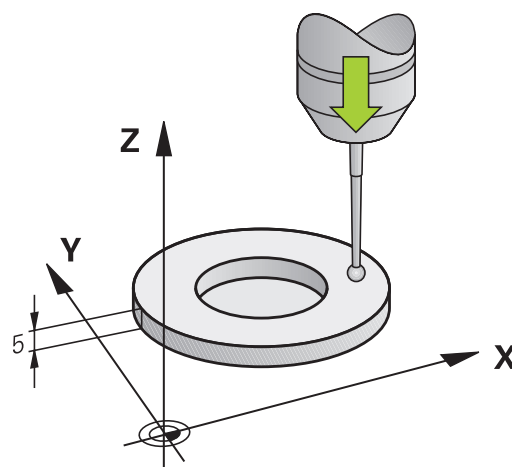
Calibrating the effective radius and compensating center misalignment

ℹ️ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

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.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

ℹ️ The center offset can be determined only with a suitable touch probe.

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.
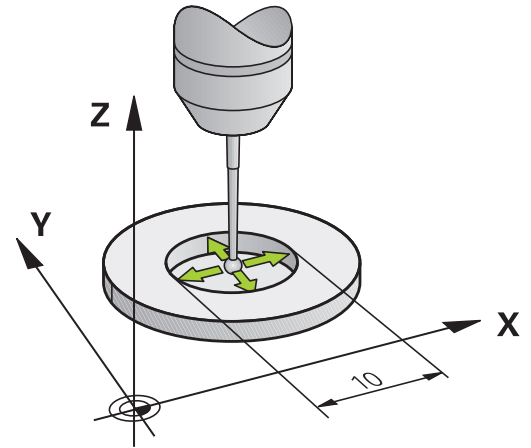
The calibration routine varies depending on how your touch probe can be oriented:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g., HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g., HEIDENHAIN touch probes with infrared transmission): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes another probing routine. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.

Calibration using a calibration ring

Proceed as follows for manual calibration using a calibration ring:

> ▶ In the **Manual operation** mode, position the ball tip inside the bore of the ring gauge

> ▶ Select the calibrating function: Press the **Calibration of a TS in a ring** soft key

> ⊳ The control displays the current calibration data.

> ▶ Enter the diameter of the ring gauge

> ▶ Probe: Press the **NC Start** key

> ⊳ The 3D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

> ▶ Check results

> ▶ Press the **OK** soft key for the values to take effect

> ▶ Press the **END** soft key to terminate the calibrating function.

> ⊳ The control logs the calibration process in the TCHPRMAN.html file.

⚙   Refer to your machine manual.

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibrating with a stud or calibration pin

Proceed as follows for manual calibration with a stud or calibration pin:
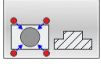
▶ In the **Manual operation** mode, position the ball tip above the center of the calibration pin

▶ Select the calibrating function: Press the **TS calibration on stud** soft key

▶ Enter the outside diameter of the stud

▶ Enter the safety clearance

▶ Probe: Press the **NC Start** key

> The 3D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

▶ Check results

▶ Press the **OK** soft key for the values to take effect

▶ Press the **END** soft key to terminate the calibrating function.

> The control logs the calibration process in the TCHPRMAN.html file.

Refer to your machine manual.

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration using a calibration sphere

Proceed as follows for manual calibration using a calibration sphere:
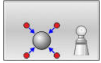
- ▶ In the **Manual operation** mode, position the ball tip above the center of the calibration sphere
- ▶ Select the calibrating function: Press the **Calibration of TS on a sphere** soft key
- ▶ Enter the outside diameter of the ball
- ▶ Enter the safety clearance
- ▶ Select Length measurement, if applicable
- ▶ If necessary, input the reference for the length
- ▶ Probe: Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- ▶ Check results
- ▶ Press the **OK** soft key for the values to take effect
- ▶ Press the **END** soft key to terminate the calibrating function
- > The control logs the calibration process in the TCHPRMAN.html file.

⚙ Refer to your machine manual.

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

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", Page 137

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.

Displaying calibration values

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 to the touch probe table in the columns **CAL_OF1** (main axis) and **CAL_OF2** (secondary axis). You can display the values on the screen by pressing the **TCH PROBE TABLE** soft key.

Further information: "Touch probe table", Page 137

During calibration, the control automatically creates the TCHPRMAN.html log file to which the calibration values are saved.

> **ⓘ** Ensure that the tool number of the tool table and the touch-probe number of the touch-probe table are correct.
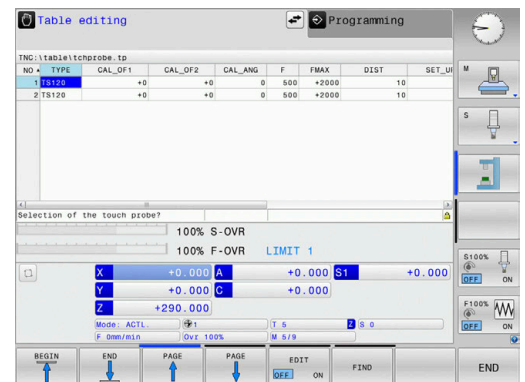
5.8 Presetting with a 3D touch probe (option number 17)

Overview

⚙ Refer to your machine manual.

The machine manufacturer can disable presetting in individual axes.

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.

ℹ HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.

The following soft-key functions are available for setting a preset on an aligned workpiece:

Soft key	Function	Page
PROBING POS	Setting a preset in any axis	203
PROBING CC	Setting a circle center as preset	204
PROBING CL	Setting the center line as preset	207

ℹ With an active datum shift, the determined value is referenced to the active preset (possibly a manual preset defined in the **Manual operation** mode). The datum shift is taken into account in the position display.

Presetting on any axis
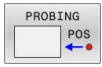
PROBING
POS

▶ Select the probing function: Press the **POSITION PROBING** soft key
▶ Move the touch probe to a position near the touch point
▶ Select the axis and probing direction, e.g. Probe in direction Z-
▶ Probe: Press the **NC Start** key
▶ **Preset**: Enter the nominal coordinate
▶ Apply with the **SET PRESET** soft key
   Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
   Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194
▶ Terminate the probing function: Press the **END** soft key
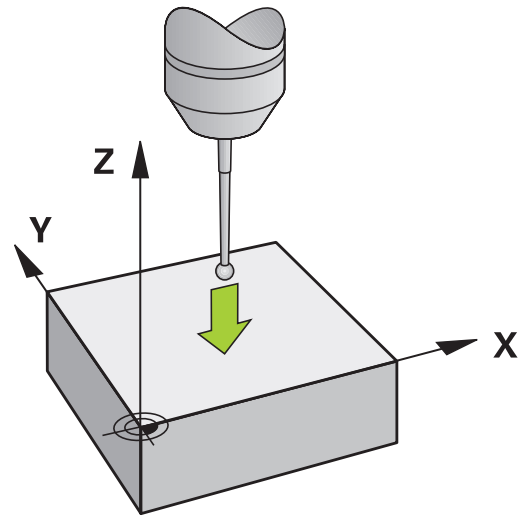
Circle center as preset

With this function, you can set the preset at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

Inside circle:

The control probes the inside wall of a circle in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

▶ Position the touch probe approximately in the center of the circle

▶ Select the probing function: Press the **PROBING CC** soft key

▶ Select the soft key for the desired probing direction

▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing process, you can have the control calculate the center (four touch points are recommended)

▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key

▶ **Preset**: Enter the X and the Y coordinates of the center of the circle in the menu window.

▶ Apply with the **SET PRESET** soft key
  Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
  Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

▶ Terminate the probing function: Press the **END** soft key

The control needs at least three touch points to calculate outside or inside circles, e.g. with circle segments. More precise results are obtained with four touch points. If possible, always pre-position the touch probe to the center.

Outside circle:
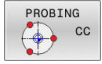
▶ Position the touch probe at a position near the first touch point outside of the circle

▶ Select the probing function: Press the **PROBING CC** soft key

▶ Select the soft key for the desired probing direction

▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected direction. Repeat this process. After the third probing process, you can have the control calculate the center (four touch points are recommended)

▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key

▶ **Preset**: Enter the coordinates of the preset

▶ Apply with the **SET PRESET** soft key
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

▶ Terminate the probing function: Press the **END** soft key
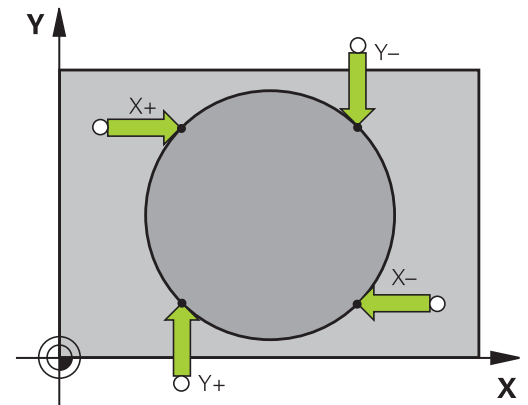
Once the probing routine is completed, the control displays the current coordinates of the circle center and the circle radius.

Setting the preset using multiple holes or circular studs

The manual **Circular pattern** probing function is part of the **Cir** probing function. Individual circles can be measured with paraxial probing operations.

A second soft-key row provides the soft key **PROBING CC (circular pattern)** for using multiple holes or circular studs to set the datum. You can set the intersection of three or more elements to be probed as a preset.

Setting the preset in the intersection of multiple holes or circular studs:

▶ Pre-position touch probe

Select the **Probing CC** probing function

▶ Select the probing function: Press the **PROBING CC** soft key

▶ Press the **PROBING CC (probing of circular pattern)** soft key

Probe a circular stud
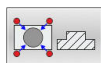
▶ Circular stud should be probed automatically: Press **Stud** soft key
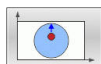
▶ Enter or select using soft key

▶ Start probing function: Press the **NC Start** key

Probe the hole.

▶ Hole should be probed automatically: Press the **Hole** soft key
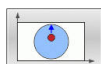
▶ Enter or select using soft key

▶ Start probing function: Press the **NC Start** key

▶ Repeat the probing procedure for the remaining elements

▶ Terminate the probing procedure and switch to the evaluation menu: Press the **EVALUATE** soft key

▶ **Preset**: Enter the X and the Y coordinates of the center of the circle in the menu window.

▶ Apply with the **SET PRESET** soft key
     Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
     Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

▶ Terminate the probing function: Press the **END** soft key

Setting a center line as preset
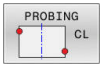
▶ Select the probing function: Press the **PROBING CL** soft key

▶ Position the touch probe at a position near the first touch point

▶ Select the probing direction by soft key

▶ Probe: Press the **NC Start** key

▶ Position the touch probe at a position near the second touch point

▶ Probe: Press the **NC Start** key

▶ **Preset**: Enter the coordinates of the preset in the menu window, confirm with the **SET PRESET** soft key, or write the value to a table
Further information: "Writing measured values from the touch probe cycles to a datum table", Page 193
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 194

▶ Terminate the probing function: Press the **END** soft key

> **i** If you desire, then after the second touch point you can change the position of the centerline in the evaluation menu, and thus the axis for setting the preset. Use the soft keys to choose between principal axis, secondary axis, and tool axis. This way you can determine the positions once, and then store them in the principal axis as well as in the secondary axis.
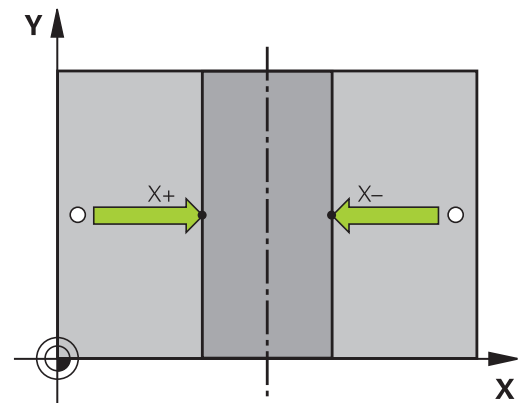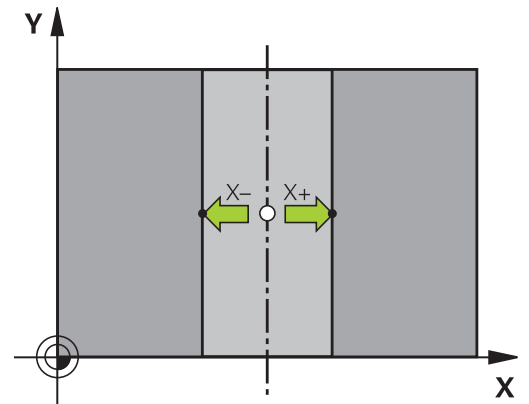
Measuring workpieces with a 3D touch probe

You can also use the touch probe in the **Manual operation** and **Electronic handwheel** operating modes to perform simple measurements on the workpiece.

With a 3D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions on the workpiece

Finding the coordinates of a position on an aligned workpiece
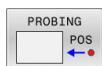
▶ Select the probing function: Press the **PROBING POS** soft key

▶ Move the touch probe to a position near the touch point

▶ Select the probing direction and the axis to which the coordinates relate: Use the corresponding soft keys to select

▶ Start the probing process: Press the **NC Start** key

> The control shows the coordinates of the touch point as preset.

Measuring workpiece dimensions
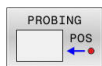
▶ Select the probing function: Press the **PROBING POS** soft key

▶ Position the touch probe at a position near the first touch point A

▶ Select the probing direction by soft key

▶ Probe: Press the **NC Start** key

▶ If you need the current preset later, write down the value that appears in the display

▶ Preset: Enter **0**.

▶ Cancel the dialog: Press the **END** key

▶ Select the probing function again: Press the **PROBING POS** soft key

▶ Position the touch probe at a position near the second touch point B

▶ Select the probe direction with the soft keys: Same axis but from the opposite direction

▶ Probe: Press the **NC Start** key

> The **Measured value** display shows the distance between the two points on the coordinate axis.
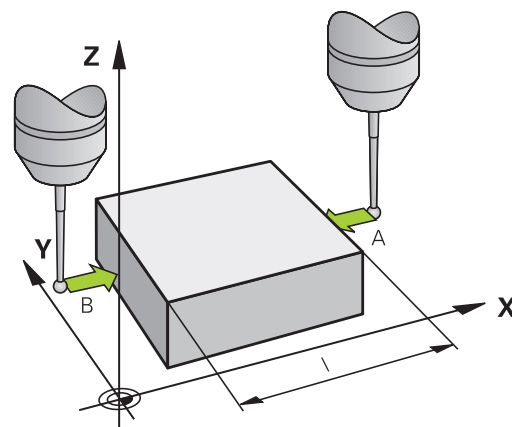
Resetting to the preset that was active before the length measurement

▶ Select the probing function: Press the **PROBING POS** soft key

▶ Probe the first touch point again

▶ Set the preset to the value that you wrote down previously

▶ Cancel the dialog: Press the **END** key

6

Testing and Running

6.1   Graphics

Application

The control graphically simulates a machining operation in the following operating modes:

- **Manual operation**
- **Program run, single block**
- **Program run, full sequence**
- **Test Run**
- **Positioning w/ Manual Data Input**

> **ℹ**   In the **Positioning w/ Manual Data Input** operating mode, the workpiece blank that is currently active in the **Program Run, Full Sequence/Single Block** operating mode is shown.

The graphic depicts the workpiece as if it were being machined with tool.

If a tool table is active, the control also takes the entries in columns **L**, **R**, **LCUTS**, **LU**, **RN**, **T-ANGLE**, **R_TIP** and **R2** into consideration.

The control will not show a graphic if

- no NC program is selected
- a screen layout without graphics was selected
- the current NC program has no valid workpiece blank definition
- with blank form definition with a subprogram, the BLK FORM block was not yet run

View options

Display the **VIEW OPTIONS** as follows:

▶ Select the desired operating mode

| VIEW
OPTIONS | ▶ Press the **VIEW OPTIONS** soft key |

The available soft keys depend on the following settings:

■ The selected view
You can select the view with the **VIEW** soft key.
■ The selected model quality
You can select the model quality in the **Graphic settings** group of the MOD menu.

The control provides the following **VIEW OPTIONS**:

Soft key	Function
WORKPIECE	Show the workpiece
TOOL	Show the tool **Further information:** "Tool", Page 213
TOOL PATHS	Show the tool paths **Further information:** "Tool", Page 213
VIEW	Select the view **Further information:** "View", Page 214
RESET TOOL PATHS	Reset the tool paths
RESET BLK FORM	Reset the workpiece blank
BLANK FRAME OFF ON	Show the workpiece blank frame
WORKPIECE EDGES OFF ON	Highlight the workpiece edges on 3D model
SHOW BLOCK NO. OFF ON	Show the block numbers of the tool paths
MARK END POINT OFF ON	Show the end points of the tool paths

Soft key	Function
WORKPIECE IN COLOR OFF ON	Show the workpiece in color
REMOVE CHIPS	Remove the chips Parts of the workpiece material that are cut off by machining will be removed from the graphics.
RESET TOOL PATHS	Reset the tool paths
	Rotate and zoom the workpiece **Further information:** "Rotating, zooming and moving a graphic", Page 215
	Shift the sectional plane in the projection in three planes **Further information:** "Shifting the sectional plane", Page 217

> **ⓘ** Operating notes:
> - The **clearPathAtBlk** machine parameter (no. 124203) allows you to specify whether the tool paths are cleared with a new BLK FORM in **Test Run** operating mode.
> - If points were output incorrectly by the postprocessor, machining marks may occur on the workpiece. To recognize these unwanted machining marks in time (prior to machining), you can test externally created NC programs for corresponding irregularities by displaying the tool paths.
> - The control saves the state of the soft keys in non-volatile memory.

Tool

Displaying the tool

If the **L** and **LCUTS** columns are defined in the tool table, the tool is graphically displayed.

> **i** A realistic tool simulation may require additional definitions (e.g., in the columns **LU** and **RN** for tool neck surfaces).

Further information: "Entering tool data into the table", Page 124

The control displays the tool in various colors:

- Turquoise: tool length
- Red: length of cutting edge and tool is engaged
- Blue: length of cutting edge and tool is retracted

Showing the tool paths

The control displays the following types of movement:

Soft key	Function
TOOL PATHS	Movements at rapid traverse and at the programmed feed rate
TOOL PATHS	Movements at the programmed feed rate
TOOL PATHS	No movements

> **i** If you move at rapid traverse inside a workpiece, both the movement and the workpiece are shown in red at the respective location.

View

The control provides the following views:

Soft key	Function
	Plan view
	Projection in three planes
	3D view

Projection in three planes

The simulation shows three sectional planes and a 3D model, similar to a technical drawing.
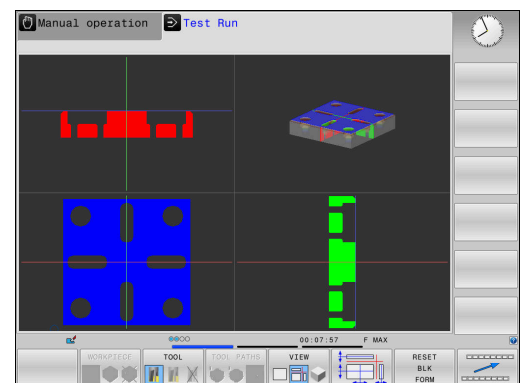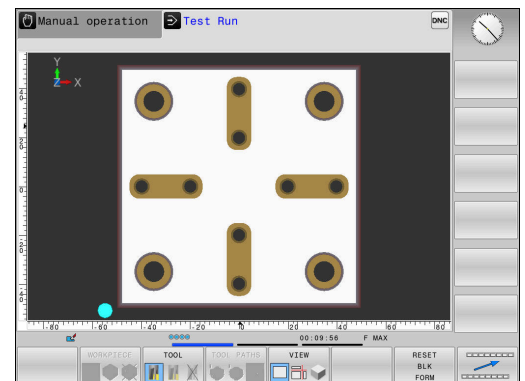
3D view

The high-resolution 3D view enables you to display the surface of the machined workpiece in greater detail. Using a simulated light source, the control creates realistic light and shadow conditions.
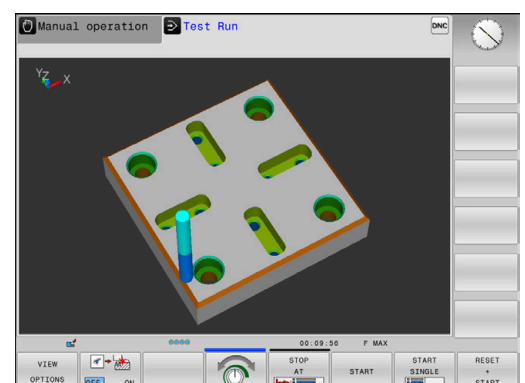
Rotating, zooming and moving a graphic

Proceed as follows to rotate a graphic, for example:
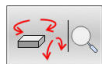
▶ Select functions for rotating and zooming

> The control displays the following soft keys.

Soft key		Function
		Rotate in 5° steps about the verti-cal axis
		Tilt in 5° steps about the horizon-tal axis
		Enlarge the graphic stepwise
		Reduce the graphic stepwise
		Reset the graphic to its original size and angle
		Move the graphic upward or downward
		Move the graphic to the left or right
		Reset the graphic to its original position and angle

You can also use the mouse to change the graphic display. The following functions are available:

▶ In order to rotate the model shown in three dimensions, hold down the right mouse button and move the mouse. If you simultaneously press the shift key, you can only rotate the model horizontally or vertically

▶ To shift the model shown: Hold the center mouse button or mouse wheel down and move the mouse. If you simultaneously press the shift key, you can only shift the model horizontally or vertically

▶ To zoom in on a certain area: Mark a zoom area by holding the left mouse button down.

> After you release the left mouse button, the control zooms in on the defined area.

▶ To rapidly magnify or reduce any area: Rotate the mouse wheel backwards or forwards

▶ To return to the standard display: Press the shift key and simultaneously double-click with the right mouse key. The rotation angle is maintained if you only double-click with the right mouse key

Setting the speed of the Test Run mode

ℹ️ The most recently set speed stays active until a power interruption. After the control has been started, the speed is set to MAX.

After you have started a program, the control displays the following soft keys with which you can set the simulation speed:

Soft key	Functions
[1:1]	Test the NC program with the speed that will be used when actually running the program (programmed feed rates will be taken into account)
[↑]	Increase the simulation speed incrementally
[↓]	Decrease the simulation speed incrementally
[MAX]	Test run at the maximum possible speed (default setting)

You can also set the simulation speed before you start a program:

[⟳]	▶ Select the function for setting the simulation speed
[↑]	▶ Select the desired function by soft key, e.g. incrementally increasing the simulation speed

Repeating graphic simulation

A part program can be graphically simulated as often as desired. To do so you can reset the graphic to the workpiece blank.

Soft key	Function
RESET BLK FORM	Show the unmachined workpiece blank

Shifting the sectional plane

The default setting of the sectional plane is selected so that it lies in the working plane in the center of the workpiece blank and in the tool axis on the top surface of the workpiece blank.

Proceed as follows to move the sectional plane:
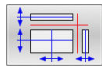
▶ Press the soft key for **shifting the sectional plane**

> The control displays the following soft keys:

Soft key		Function
		Shift the vertical sectional plane to the right or left
		Shift the vertical sectional plane forward or backward
		Shift the horizontal sectional plane upwards or downwards

The position of the sectional planes is visible during shifting. The shift remains active, even if you activate a new workpiece blank.

Resetting sectional planes

The shifted sectional plane also remains active for a new workpiece blank. The sectional plan is automatically reset when the control is restarted.

Proceed as follows to move the sectional plane to the default position:
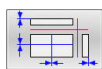
▶ Press the soft key for **resetting the sectional planes** soft key

6.2 Measurement of machining time

Application

Machining time in the Test Run operating mode

The control calculates the duration of the tool movements and displays this as machining time in the test run. The control takes feed movements and dwell times into account.

The control does not dwell during the program test, but adds the dwell times to the machining time.

The time determined by the control is only of limited value for calculating the machining time because it does not take any machine-dependent time intervals (e.g., for tool changes) into consideration.

Proceed as follows to select the stopwatch function:

▶ Select the stopwatch function

▶ Select the desired function by soft key, e.g. saving the displayed time

Soft key	Stopwatch functions
STORE	Save the displayed time
ADD	Display the sum of saved time and displayed time
RESET 00:00:00	Clear the displayed time

Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

6.3 Showing the workpiece blank in the working space

Application

In **Test Run** operating mode, you can graphically check the position of the workpiece blank and the preset in the working space of the machine. The graphics show the preset that has been set in the NC program using Cycle **247**. If you have not defined a preset in the NC program, then the graphic shows the active preset on the machine.

A transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The control takes over the dimensions from the workpiece blank definition of the selected NC program.

For a test run it normally does not matter where the workpiece blank is located within the working space. If you activate **BLANK IN WORK SPACE** working space monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

In addition, you can use the current machine status for the **Test Run** operating mode.

The current machine status includes the following:

- active machine kinematics
- active traverse ranges
- active machining modes
- active working spaces
- active preset
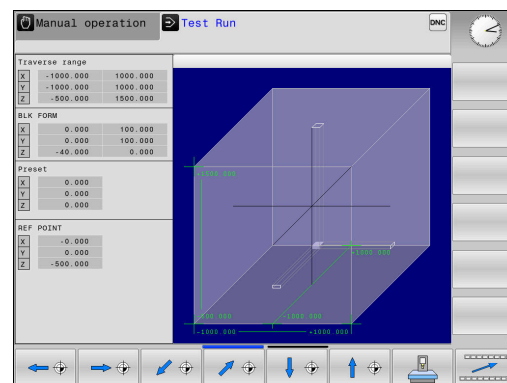
Soft key		Function
←⊕	→⊕	Shift the workpiece blank in positive or negative X direction
↙⊕	↗⊕	Shift the workpiece blank in positive or negative Y direction
↑⊕	↓⊕	Shift the workpiece blank in positive or negative Z direction
⬚		Use the current machine status
ACTIVE TRAVERSE RANGES		Display the current traverse range
SELECT TRAVERSE RANGE		Selecting the traverse range. The ranges of traverse are configured by the machine manufacturer.

Soft key	Function
SW LIMIT MONITORING OFF [ON]	Switch on/off the monitoring function
MACHINE REF POINT OFF [ON]	Display the machine preset
RESET THE PRESET	Set the principal axis values of the active preset to 0 for the simulation

> For blank in work space, the control depicts the **BLK FORM** only schematically.
>
> ■ With **BLK FORM CYLINDER**, a cuboid is depicted as the workpiece blank

6.4 Measuring

Application

In the **Test Run** operating mode, you can use the **MEASURING** soft key to display the following information:

- Approximated coordinates as XYZ values with reference to workpiece coordinate system **W-CS**
  Further information: "Workpiece coordinate system W-CS", Page
- Optional display
  - FMAX: If the control machines at the maximum feed rate.
- Tool number
- Tool name

Proceed as follows to select the measuring function:

▶ Set the **MEASURING** soft key to **ON**

▶ Position the mouse pointer on the respective position
> The control displays the positioning ball icon and the surface orientation with a black and white circle and a perpendicular line on it.
> The corresponding information is displayed in a blue text field.

ℹ The **MEASURING** soft key is available in the following views:
- Plan view
- 3D view
Further information: "View", Page 214

6.5   Optional program-run interruption

Application

Refer to your machine manual.

The behavior of this function varies depending on the respective machine.

The control optionally interrupts program run at NC blocks in which an M1 has been programmed. If you use M1 in the **Program run** operating mode, the control does not switch off the spindle or the coolant.

▶ Set the **M01** soft key to **OFF**
> The control does not interrupt the **Program run** or **Test Run** at NC blocks that contain an M1.

▶ Set the **M01** soft key to **ON**
> The control interrupts the **Program run** or **Test Run** at NC blocks with M1.

6.6 Skipping NC blocks

You can have NC blocks skipped in the following operating modes:

- **Test Run**
- **Program run, full sequence**
- **Program run, single block**
- **Positioning w/ Manual Data Input**

> **ℹ** Operating notes:
> - This function does not work for **TOOL DEF** blocks.
> - After a power interruption the control returns to the most recently selected setting.
> - The setting of the **HIDE** soft key is only effective in the respective operating mode.

Test Run and Program Run

Application

In **Test Run** or **Program Run, Full Sequence/Single Block** operating mode, the control can skip over NC blocks that you have preceded by a slash (**/**):

HIDE ON OFF	▶ Set the **HIDE** soft key to **ON**
	> The control skips the NC blocks.
HIDE ON OFF	▶ Set the **HIDE** soft key to **OFF**
	> The control runs or tests the NC blocks.

Procedure

You can optionally hide NC blocks.

To hide NC blocks in the **Programming** mode:

↓	▶ Select the desired NC block

INSERT REMOVE	▶ Press the **INSERT** soft key
	> The control inserts a slash (/).

To show NC blocks again in the **Programming** mode:

↓	▶ Select the hidden NC block

INSERT REMOVE	▶ Press the **REMOVE** soft key
	> The control removes the slash (/).

Positioning w/ Manual Data Input

Application

> 🛈   In order to skip NC blocks in the **Positioning w/ Manual Data Input** operating mode, you need an alphabetic keyboard.

In **Positioning w/ Manual Data Input** operating mode, you can have the control skip NC blocks that have been marked correspondingly. Proceed as follows:

> ▶   Set the **HIDE** soft key to **ON**
> >   The control skips the NC blocks.
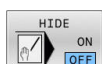
> ▶   Set the **HIDE** soft key to **OFF**
> >   The control runs the NC blocks.

Procedure

To hide NC blocks in the **Positioning w/ Manual Data Input** operating mode:

> ▶   Select the desired NC block

> ▶   Press the **/** key on the alphabetic keyboard
> >   The control inserts a slash (**/**).

To show NC blocks again in the **Positioning w/ Manual Data Input** operating mode:

> ▶   Select the hidden NC block

> ▶   Press the **Backspace** key
> >   The control removes the slash (**/**).

6.7 Test run

Application

The simulation of NC programs and program sections in **Test Run** operating mode helps you to detect programming errors, as well as collisions and interruptions during program run, before the actual machining process. The motion simulation enables the visual inspection of both the machining result and the machine movements.

The control helps you detect the following problems:

- Programming errors
  - Geometrical incompatibilities
  - Missing data
  - Impossible jumps
- Machining errors
  - Usage of locked tools
  - Violation of the machine's working space

The following functions and information are available:

- Block-by-block simulation
- Cancellation of test at any NC block
- Hiding or skipping of NC blocks
- Determined machining time
- Additional status display
- Graphic display

> **i** The graphical representation functions and the quality of the represented model depend on the settings in the **Graphic settings** MOD function.
>
> **Further information:** "Graphic settings", Page 272

Keep the following in mind when performing a test run

With cuboid workpiece blanks, the control starts a test run after a tool call at the following position:

- In the working plane in the center of the defined **BLK FORM**
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

The **FN 27: TABWRITE** and **FUNCTION FILE** functions are only considered in the **Program run, single block** and **Program run, full sequence** operating modes.

NOTICE

Danger of collision!

In the **Test Run** operating mode, the control does not take all axis movements of the machine into consideration (e.g., PLC positioning movements as well as movements from tool-change macros and M functions). Thus, the test might be performed without errors, but later operation will deviate from these results. Danger of collision during machining!

- ▶ Test the NC program at the later machining position (**BLANK IN WORK SPACE**)
- ▶ Program a safe intermediate position after the tool change and before prepositioning
- ▶ Carefully test the NC program in the **Program run, single block** operating mode

---

NOTICE

Danger of collision!

If you simulate an NC program that includes SQL commands, the control might overwrite table values. Overwriting table values might result in incorrect positioning of the machine. There is a danger of collision.

- ▶ Program NC programs in such a way that SQL commands are not executed during simulation
- ▶ Use **FN18: SYSREAD ID992 NR16** to check whether the NC program is active in a different operating mode or in **Simulation**

⚙   Refer to your machine manual.

Your machine manufacturer can also define a tool-change macro for the **Test Run** operating mode. This macro will simulate the exact behavior of the machine.

In doing so, the machine manufacturer often changes the simulated tool change position.

Test run execution

> ⓘ For the test run, you must activate a tool table (status S). Select a tool table via the file manager in the **Test Run** operating mode.

You can select any preset table (status S) for the test run.

Once you press the **RESET + START** soft key in the **Test Run** operating mode, the control automatically uses the active preset from the machine operating modes for simulation. This preset remains selected when starting the test run until you define another preset in the NC program. The control reads all other defined presets from the preset table selected for the test run.

With the **BLANK IN WORK SPACE** function, you can activate workspace monitoring for the test run.

Further information: "Showing the workpiece blank in the working space ", Page 219

| ⇥ | ▶ Operating mode: Press the **Test Run** key |

| PGM MGT | ▶ Call the file manager with the **PGM MGT** key and select the file you wish to test |

The control then displays the following soft keys:

Soft key	Function
RESET + START	Reset the blank form, reset the previous tool data and test the entire NC program
START	Test the entire NC program
START SINGLE	Test each NC block individually
STOP AT	Executes the **Test Run** until NC block N
STOP	Stop test run (this soft key only appears if you have started the test run)

You can interrupt and continue the test run at any time, even within machining cycles. In order to continue the test, the following actions must not be performed:

- Selecting another NC block with the arrow keys or the **GOTO** key
- Making changes to the NC program
- Selecting another NC program

Executing a Test Run up to a certain NC block

With the **STOP AT** function the control executes a **Test Run** up to the NC block with block number **N**.

To stop the **Test Run** at the desired NC block:

▶ Press the **STOP AT** soft key

▶ **Stop at: N =** Enter the block number at which the simulation should stop

▶ Enter the **Program** name of the NC program containing the NC block with the selected block number

> The control shows the name of the selected NC program.

▶ If the simulation is to be stopped in an NC program that has been called using, for example, **CALL PGM**, then enter this name.

▶ **Repetitions =** If **N** is located in a program-section repeat, enter the number of repeats that you want to run.
   Default 1: The control stops before **N** is simulated
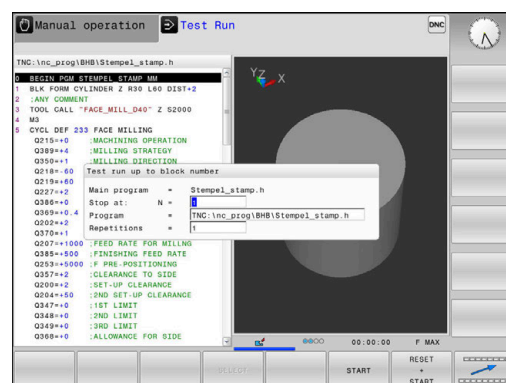
Possibilities in stopped condition

If you interrupt the **Test Run** with the **STOP AT** function, you have the following possibilities in this stopped condition:

- Enable or disable **NC block skip**
- **Optional program stop** enable or disable
- Modify graphics resolution and model
- Modify the NC program in the **Programming** operating mode

If you modify the NC program in the **Programming** operating mode, the simulation behaves as follows:

- Modification before the interruption point: The simulation restarts at the beginning
- Modification after the interruption point: Positioning at the interruption point is possible with **GOTO**

Using the GOTO key

Jumping with the GOTO key

Use the **GOTO** key to jump to a specific location in the NC program, regardless of the active operating mode.

Proceed as follows:

GOTO ☐	▶ Press the **GOTO** key
	> The control opens a pop-up window.
	▶ Enter a number
N LINES ↓	▶ Select the jump statement by soft key, e.g. move down the number of lines entered
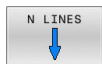
The control provides the following options:

Soft key	Function
N LINES ↑	Move up the number of lines entered
N LINES ↓	Move down the number of lines entered
GOTO LINE NUMBER	Jump to the block number entered

> ℹ️ Use the **GOTO** function only during programming and testing of NC programs. Use the **Block scan** function during program run.
>
> **Further information:** "Starting the NC program at any point: Block scan", Page 242

Quick selection with the GOTO key

With the **GOTO** key, you can open the Smart Select window that makes it easy for you to select special functions or cycles.

To select special functions:

SPEC FCT	▶ Press the **SPEC FCT** key
GOTO ☐	▶ Press the **GOTO** key
	> The control displays a pop-up window showing a structural view of the special functions
	▶ Select the desired function

Further information: User's Manual for **Klartext Programming**

Opening the selection window with the GOTO key

When the control provides a selection menu, you can use the **GOTO** key to open the selection window. This allows you to view the available entries.

Scrollbar

Screen content can be shifted with the mouse using the scroll bar at the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.

6.8 Program run

Application

In the **Program run, full sequence** operating mode, the control executes an NC program continuously to its end or up to a program stop.

In the **Program run, single block** operating mode, the control executes each NC block individually after pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point. The workpiece blank definition will be interpreted as a separate NC block.

You can use the following control functions in the **Program run, single block** and **Program run, full sequence** operating modes:

- Interrupt program run
- Start the program run from a certain NC block
- Skip NC blocks
- Edit the tool table TOOL.T
- Edit the active datum table or compensation table
- Checking and changing Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display
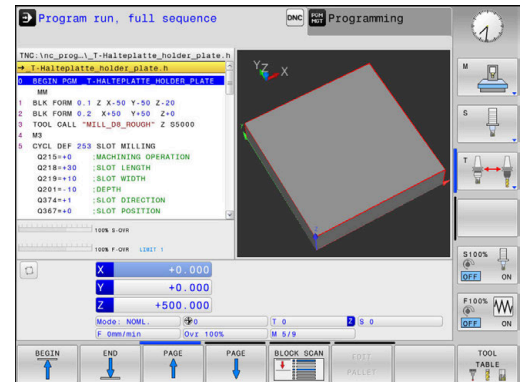
> ### NOTICE
>
> **Caution: Danger due to manipulated data!**
>
> If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.
>
> ▶ Copy the NC program and all called files to the **TNC:** drive

---

Running an NC program

Preparation

▶ Clamp the workpiece to the machine table

▶ Set a preset

▶ Select the necessary tables (status M)

▶ Select the NC program (status M)

> **i** Operating notes:
> - You can change the feed rate and spindle speed using the potentiometers.
> - You can reduce the feed rate using the **FMAX** soft key. This reduction affects all rapid traverse and feed movements, even after the control has been restarted.

Program Run, Full Sequence

▶ Start the NC program with the **NC Start** key

Program Run, Single Block

▶ Start each NC block of the NC program individually with the **NC Start** key

Structuring NC programs

Definition and applications

The control enables you to comment NC programs in structuring blocks. Structuring blocks are texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex NC programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the NC program later. Structuring blocks can be inserted into the NC program at any point.

Structure blocks can also be displayed in a separate window, and be edited or added to, as desired. Use the appropriate screen layout for this.

The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Programming**

Displaying the program structure window / Changing the active window

▶ Display structure window: For this screen layout press the **PROGRAM + SECTS** soft key

▶ Change the active window: Press the **CHANGE WINDOW** soft key

Selecting blocks in the program structure window

If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.
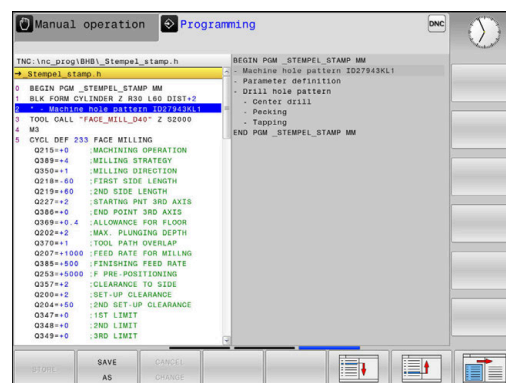
Checking and changing Q parameters

Procedure

You can check Q parameters in all operating modes, and also edit them.

▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC STOP** key and the **INTERNAL STOP** soft key) or stop the test run

Q
INFO
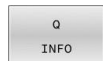
▶ To call the Q parameter functions, press the **Q INFO** soft key or the **Q** key

> The control lists all of the parameters and their corresponding current values.

▶ Use the arrow keys or the **GOTO** key to select the desired parameter.

▶ If you want to change the value, then press the **EDIT CURRENT FIELD** soft key, enter the new value, and confirm with the **ENT** key

▶ If you want to leave the value unchanged, then press the **PRESENT VALUE** soft key or close the dialog with the **END** key

ⓘ If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while program run has been interrupted or aborted.

This status is reached after an NC block has been executed, for example in **Program run, single block**

The following Q and QS parameters cannot be edited in the **Q parameter list** window:

■ Variable range from 100 to 199, because there might be interferences with special functions in the control.

■ Variable range from 1200 to 1399, because there might be interferences with machine manufacturer-specific functions.

All of the parameters with displayed comments are used by the control within cycles or as transfer parameters.
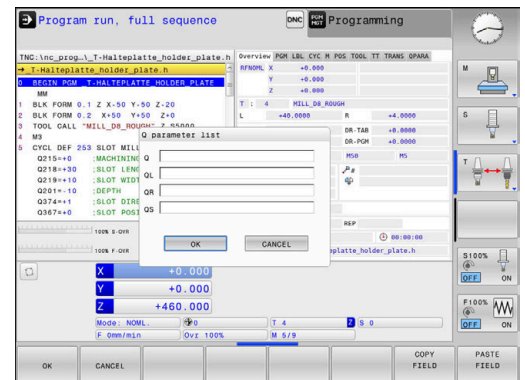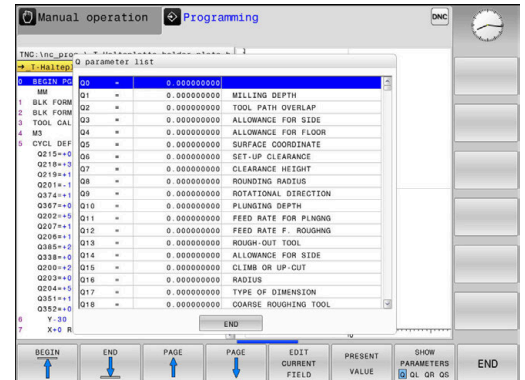
You can have Q parameters also be displayed in the additional status
display in all operating modes (except **Programming** mode).

▶ If needed, interrupt the program run (e.g., by pressing the
   NC STOP key and the **INTERNAL STOP** soft key), or stop the test
   run
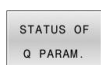
| ○ | ▶ Display the soft key row for screen layout |

| PROGRAM + STATUS | ▶ Select the layout option for the additional status display |
| | > In the right half of the screen, the control shows the **Overview** status form. |

| STATUS OF Q PARAM. | ▶ Press the **STATUS OF Q PARAM.** soft key |

Q PARAMETER LIST	▶ Press the **Q PARAMETER LIST** soft key.
	> The control opens a pop-up window.
	▶ For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters

| ⓘ | The display in the **QPARA** tab always contains eight decimal places. The result of **Q1 = COS 89.999** is shown by the control as 0.00001745, for example. Very large or very small values are displayed by the control in exponential notation. The result of **Q1 = COS 89.999 * 0.001** is shown by the control as +1.74532925e-08, where e-08 corresponds to the factor of 10^{-8}. |

Interrupting, stopping or canceling a program

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M0**)
- Stop the program run (e.g., with the **NC Stop** key)
- Stop the program run (e.g., with the **NC Stop** key in connection with the **INTERNAL STOP** soft key)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

The control shows the current program run status in the status display.

Further information: "General status display", Page 68

In contrast to a stopped run, an interrupted, aborted (terminated) program run enables certain actions by the user, including the following:

- Selecting an operating mode
- 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 **/**

> **i** Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Program-controlled 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 **M0**
- Conditional stop **M1**

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the **Q INFO** soft key
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

---

Manual program interruption

While an NC program is being executed in the **Program run, full sequence** operating mode, select the **Program run, single block** operating mode. The control interrupts the machining process at the end of the current machining step.

Aborting program run

▶ Press the **NC Stop** key
> The control does not finish the current NC block.
> The control shows the icon for stopped status in the status display.
> Actions such as a change of operating mode are not possible.
> The program can be resumed with the **NC Start** key.

▶ Press the **INTERNAL STOP** soft key

> The control briefly shows the icon for program cancellation in the status display.
> The control shows the icon for the exited inactive status in the status display.
> Actions such as a change of operating mode are available again.

Compensations during program run

Application

During the program run, you can access the programmed compensation tables and the active datum table. You can also make changes to these tables. The changes only become effective after activating the compensation again.

Functionality

A datum table can be activated by using the **SEL TABLE** function within an NC program. The datum table remains active until you select a new one.

Further information: User's Manual for **Klartext Programming**

The control displays the following information on the **TRANS** tab of the additional status display:

- Name and path of the active datum table
- Active datum number
- Comment from the **DOC** column of the active datum number

Correction tables can be activated by using the **SEL CORR-TABLE** function in the NC program.

Further information: User's Manual for **Klartext Programming**

Activating tables manually

> ⓘ If you work without **SEL TABLE,** you must activate the
> desired datum table or compensation table in the **Program
> run, single block** or **Program run, full sequence**
> operating mode.

To activate a table in the **Program run, full sequence** operating
mode:

⊡ ▶ Change to the **Program run, full sequence**
operating mode.

PGM
MGT ▶ Press the **PGM MGT** key

▶ Select the desired table

> The control activates the table for program run
and marks the file with the **M** status.

Editing a compensation table during program run

To edit a compensation table in the Program Run operating mode:

SELECT
COMPENS.
TABLES ▶ Press the **OPEN COMPENS. TABLES** soft key

DATUM
TABLE ▶ Press the soft key for the desired table (e.g.,
DATUM TABLE

> The control opens the active datum table.

EDIT
OFF ☐ON ▶ Set the **EDIT** soft key to **ON**

▶ Select the desired value

▶ Edit the value

> ⓘ The changed data do not take effect until after the
> compensation has been activated again.

Transferring the actual position into the datum table

In the datum table, you can enter the current tool position in the
respective axis by using the **ACTUAL POSITION CAPTURE** key.

The current tool position can be transferred into the datum table as
follows:
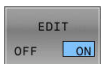
EDIT
OFF ☐ON ▶ Set the **EDIT** soft key to **ON**

▶ Select the desired value

⊹ ▶ Press the **ACTUAL POSITION CAPTURE** key

> The control now loads the current position into
the selected axis.

> ⓘ After you have changed a value in a datum table, you must
> save the change with the **ENT** key. Otherwise, the change
> will not be taken into account when the NC program is
> executed.
>
> A datum change will only become effective after you have
> called Cycle **7** or **TRANS DATUM** again.

Moving the machine axes during an interruption

During an interruption, you can move the machine axes in the same way as in the **Manual Operation** mode.

Modifying the preset during an interruption

If you modify the active preset during an interruption, resuming the program run is only possible with **GOTO** or mid-program startup at the interruption point.

Example: Retracting the spindle after tool breakage

▶ Interrupt machining
▶ Enable the axis-direction keys: Press the **MANUAL TRAVERSE** soft key
▶ Move the machine axes with the axis direction keys

⊚ Refer to your machine manual.
   On some machines, you may have to press the **NC Start** key after the **MANUAL TRAVERSE** soft key to enable the axis-direction keys.

Resuming program run after an interruption

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour after manual machine axis positioning during an interruption (**RESTORE POSITION** soft key).

ℹ Operating notes:
- The saved data remains active until it is reset (e.g., by selecting a program).
- If you interrupt an NC program using the **INTERNAL STOP** soft key, you have to start machining at the start of the program or by using the **BLOCK SCAN** function.
- For program interruptions within program-section repeats or subprograms, you have to use the **BLOCK SCAN** function for re-entering at the point of interruption.
- With machining cycles, mid-program startup is always executed at the start of the cycle. If you interrupt a program run during a machining cycle, the control repeats machining steps already carried out after a block scan.

Resuming the program run with the NC Start key

You can resume program run by pressing the **NC Start** key if the NC program was stopped in one of the following ways:

- Press the **NC Stop** key
- Programmed interruption

Resuming program run after an error

With an erasable error message:

▶ Remove the cause of the error

▶ Clear the error message from the screen: Press the **CE** key

▶ Restart the program, or resume program run where it was interrupted

Retraction after a power interruption

> ⚙ Refer to your machine manual.
>
> Your machine manufacturer configures and enables the **Retraction** mode of operation.

With the **Retraction** mode of operation you can disengage the tool from the workpiece after an interruption in power.

If you activated a feed rate limit before a power failure, this is still active. You can deactivate the feed-rate limit using the **CANCEL THE FEED RATE LIMITATION** soft key.

The **Retraction** mode of operation is selectable in the following conditions:

■ Power interrupted

■ No control voltage for the relay

■ Traversing the reference points

The **Retraction** operating mode offers the following modes of traverse:

Mode	Function
Machine axes	Movement of all axes in the machine coordinate system
Thread	Movements of the tool axis in the active coordinate system with compensating movement of the spindle
	Effective parameters: Thread pitch and direction of rotation

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.

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 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 will then display the **Power interrupted** message in the screen header.

▶ Activate the **Retraction** mode: Press the **RETRACT** soft key
> The control displays the message **Retraction selected**

▶ Confirm the power interruption: Press the **CE** key
> The control compiles the PLC program.

▶ Switch on the machine control voltage
> The control checks the functioning of the EMERGENCY STOP circuit. If there is at least one non-referenced axis, you will have to compare the displayed position values with the actual axis values and confirm that they are correct. If required, follow the dialog.

▶ Check the preselected traverse mode: If required, select **THREAD**
▶ Check the preselected thread pitch: If required, enter the thread pitch
▶ Check the preselected direction of rotation: if needed, select the turning direction of the thread
Right-handed thread: the main spindle turns clockwise when moving into the workpiece, counter-clockwise when retracting from it; left-handed thread: main spindle turns counter-clockwise when moving into the workpiece and clockwise when retracting from it

RETRACT ▶ Activate retraction: Press the **RETRACT** soft key

▶ Retraction: Retract the tool with the axis direction keys or the electronic handwheel
Axis key Z+: Retraction from the workpiece
Axis key Z-: Moving into the workpiece

▶ Exit retraction: Return to the original soft-key level

END RETRACTION ▶ End the **Retraction** operating mode: Press the **END RETRACTION** soft key
> The control checks whether the **Retraction** operating mode can be ended. If necessary, follow the dialog.

▶ Answer confirmation request: If the tool was not correctly retracted, press the **NO** soft key. If the tool was correctly retracted, press the **YES** soft key.
> The control hides the **Retraction selected** dialog.
▶ Initialize the machine: If required, traverse the reference points
▶ Establish the desired machine condition

Starting the NC program at any point: Block scan

⚙ Refer to your machine manual.
The **BLOCK SCAN** function must be enabled and configured by the machine manufacturer.

The **BLOCK SCAN** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations.

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- The **INTERNAL STOP** soft key
- 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.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary
- Multi-level block scan in subprograms and touch probe cycles
- Block scan in a point table

At the start of the block scan, the control resets all data, as when selecting an NC program. During the block scan, you can switch between **Program Run Full Sequence** and **Program Run Single Block**.
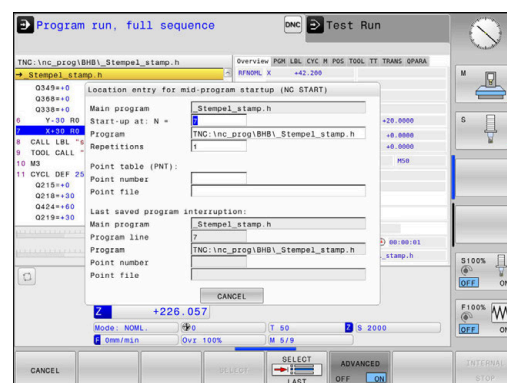
NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

▶ Use the **Block scan** function in multiple steps

Procedure for simple block scan

ℹ The control only displays the dialogs required by the process in the pop-up window.
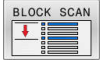
▶ Press the **BLOCK SCAN** soft key
> The control shows a pop-up window with the active main program.
▶ **Start-up at: N =**: Enter the number of the NC block where you wish to enter the NC program
▶ **Program**: Check the name and path of the NC program containing the NC block, or enter with the **SELECT** soft key
▶ **Repetitions**: If the NC block is located within a program section repeat, enter the number of the repetition that should be run next.

▶ Press the **ADVANCED** soft key if needed

▶ If needed, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption

▶ Press the **NC Start** key
> The control starts the block scan, calculates until the entered NC block, and shows the next dialog.

If you changed the machine status:

▶ Press the **NC Start** key
> The control restores the machine status (e.g., TOOL CALL or M functions and shows the next dialog).

If you changed the axis positions:

▶ Press the **NC Start** key
> The control approaches the specified positions in the specified sequence and shows the next dialog.
Approach axes in individually selected sequence:
Further information: "Returning to the contour", Page 247

▶ Press the **NC Start** key
> The control resumes execution of the NC program.

Example of simple block scan

i Even after an internal stop, the control shows the number of repetitions on the Overview tab of the status display.

After an internal stop, you would like to start in NC block 12 in the third machining operation of LBL 1.

Enter the following data in the pop-up window:

- **Start-up at: N =**12
- **Repetitions** 3

Procedure for 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 purpose, jump in the main program to the desired subprogram call. With the **CONTINUE BLOCK SCAN** function, you can jump further from this position.

i Operating notes:
- The control only displays the dialogs required by the process in the pop-up window.
- You can also continue the **BLOCK SCAN** without restoring the machine status and the axis position of the first startup point. For this, press the **CONTINUE BLOCK SCAN** soft key before confirming the restoration with the **NC Start** key.

Block scan to the first start-up point:
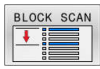
	▶ Press the **BLOCK SCAN** soft key
	▶ Enter the first NC block where you wish to start
	▶ Press the **ADVANCED** soft key if needed
	▶ If needed, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption
	▶ Press the **NC Start** key
	> The control starts the block scan and calculates until the entered NC block.
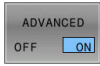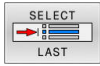
If the control should restore the machine status of the entered NC block:

| | ▶ Press the **NC Start** key |
| | > The control restores the machine status (e.g., TOOL CALL, M functions). |

If the control should restore the axis positions:

| | ▶ Press the **NC Start** key |
| | > The control moves in the specified sequence to the specified positions. |

If the control should run the NC block:

	▶ Select the **Program Run Single Block** operating mode if required
	▶ Press the **NC Start** key
	> The control runs the NC block.

Block scan to the next start-up point:
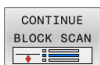
| | ▶ Press the **CONTINUE BLOCK SCAN** soft key |
| | ▶ Enter the NC block where you wish to start |

If you changed the machine status:

| | ▶ Press the **NC Start** key |

If you changed the axis positions:

| | ▶ Press the **NC Start** key |

If the control should run the NC block:

	▶ Press the **NC Start** key
	▶ Repeat these steps if required to jump to the next start-up point
	▶ Press the **NC Start** key
	> The control resumes execution of the NC program.

Example of multi-level block scan

You run a main program with several subprogram calls in the NC program Sub.h. You work with a touch probe cycle in the main program. You use the result of the touch probe cycle later for positioning.

After an internal stop you wish to start up in NC block 8 in the second call of the subprogram. This subprogram call is in NC block 53 of the main program. The touch probe cycle is in NC block 28 of the main program (i.e., before the desired start-up point).
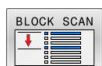
▶ Press the **BLOCK SCAN** soft key
▶ In the pop-up window enter the following data:
  ■ **Start-up at: N =**28
  ■ **Repetitions** 1

▶ Select the **Program Run Single Block** operating mode if required

▶ Press the **NC Start** key until the control runs the touch probe cycle
> The control saves the result.
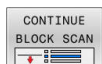
▶ Press the **CONTINUE BLOCK SCAN** soft key
▶ In the pop-up window enter the following data:
  ■ **Start-up at: N =**53
  ■ **Repetitions** 1

▶ Press the **NC Start** key until the control runs the NC block
> The control jumps into the subprogram Sub.h.
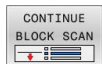
▶ Press the **CONTINUE BLOCK SCAN** soft key
▶ In the pop-up window enter the following data:
  ■ **Start-up at: N =**8
  ■ **Repetitions** 1

▶ Press the **NC Start** key until the control runs the NC block
> The control continues to run the subprogram and then returns to the main program.

Block scan in a point table

If you start in a point table called by the main program, use the **ADVANCED** soft key.
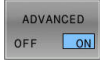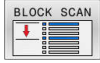
[BLOCK SCAN icon]	▶ Press the **BLOCK SCAN** soft key
	> The control shows a pop-up window.
[ADVANCED OFF ON icon]	▶ Press the **ADVANCED** soft key
	> The control expands the pop-up window.
	▶ **Point number**: Enter the row number of the point table you start with
	▶ **Point file**: Enter the name and path of the point table
[SELECT LAST icon]	▶ If required, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption
[NC Start icon]	▶ Press the **NC Start** key

If you would like to use the block scan function to start in a point pattern, then proceed just as you would for starting in the point table. Enter the desired point number in the **Point number** input field. The first point in the point pattern has the point number **0**.

Returning to the contour

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
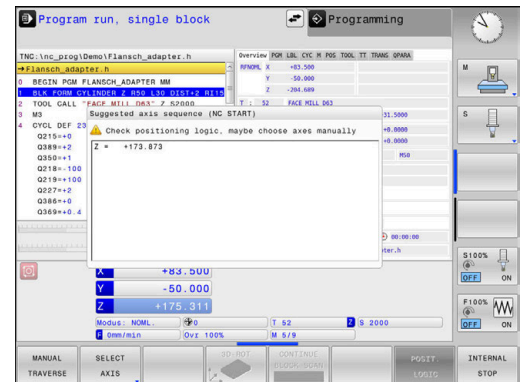
Procedure

To approach the contour:

> RESTORE POSITION

▶ Press the **RESTORE POSITION** soft key

▶ Restore the machine status, if required

Approach the axes in the sequence shown by the control:

> [NC Start icon]

▶ Press the **NC Start** key

Approach the axes according to individually selected sequence:

> SELECT AXIS

▶ Press the **SELECT AXIS** soft key

▶ Press the soft key of the first axis

> [NC Start icon]

▶ Press the **NC Start** key

▶ Press the soft key of the second axis

> [NC Start icon]

▶ Press the **NC Start** key

▶ Repeat the process for all axes

> ℹ️ 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.

Procedure for manual axes

Manual axes are non-driven axes that need to be positioned by the machine operator.

If the axes to be returned to the contour include manual axes, the control does not display a sequence for approaching. The control automatically displays the soft keys of the available axes.

To return to the contour:

> W

▶ Press the soft key of the manual axis

▶ Position the manual axis to the value shown in the dialog

> When a manual axis with encoder has reached the position, the control automatically removes the value from the dialog.

> Axes W in position

▶ Press the soft key of the manual axis again

> The control saves the position.
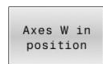
> ℹ️ When all manual axes available have been positioned, the control suggest a sequence for positioning the remaining axes.
>
> In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

6.9 Functions for program display

Overview

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control displays the following soft keys for displaying the NC program in pages:

Soft key	Function
PAGE ⬆	Go back one screen in the NC program
PAGE ⬇	Go forward one screen in the NC program
BEGIN ⬆	Select start of program
END ⬇	Select end of program

6.10 Positioning w/ Manual Data Input operating mode

The **Positioning w/ Manual Data Input** operating mode is particularly convenient for simple machining operations or for pre-positioning the tool. It enables you to write a short NC program in Klartext format and execute it immediately. The NC program is stored in a file named $MDI.

You can use the following functions for example:

- Cycles
- Radius compensation
- Program section repetitions
- Q parameters

The additional status display can be activated in the **Positioning w/ Manual Data Input** operating mode.

Positioning with manual data input (MDI)

▶ Select **Positioning w/ Manual Data Input** operating mode

▶ Program the desired available function

▶ Press the **NC Start** key

> The control executes the highlighted NC block.
> **Further information:** "Positioning w/ Manual Data Input operating mode", Page 250

i Operating and programming notes:

- The following functions are not available in the **Positioning w/ Manual Data Input** operating mode:
  - Program call
    - **CALL PGM**
    - **SEL PGM**
    - **CALL SELECTED PGM**
  - Programming graphics
  - Program-run graphics
- Using the **SELECT BLOCK**, **CUT OUT BLOCK** etc. soft keys you can also conveniently and rapidly reuse program sections from other NC programs.
  Further information: User's Manual for Klartext Programming
- You can control and modify Q parameters with the **Q PARAMETER LIST** and **Q INFO** soft keys.
  Further information: "Checking and changing Q parameters", Page 233

Example

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the preset, you can program and execute the drilling operation with a few lines of programming.

First you pre-position the tool above the workpiece with straight-line blocks and position with a safety clearance of 5 mm above the hole. Then drill the hole with Cycle **200 DRILLING**.
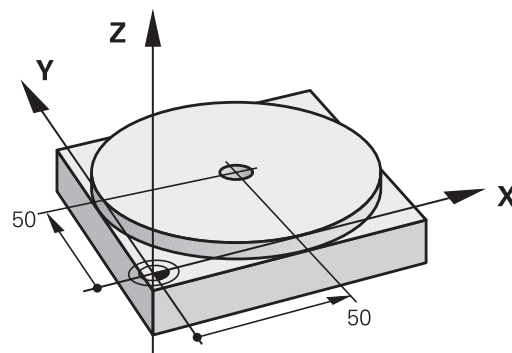
NC block	Description
0 BEGIN PGM $MDI MM	
1 TOOL CALL 1 Z S2000	Call the tool: tool axis Z,
	spindle speed 2000 rpm
2 Z+200 R0 FMAX	Retract tool (F MAX = rapid traverse)
3 Y+50 R0 FMAX M3	Move the tool at F MAX to a position above the hole, spindle on
4 X+50 R0 FMAX	Move the tool at F MAX to a position above the hole
5 CYCL DEF 200 DRILLING	Define the DRILLING cycle
Q200=5 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-20 ;DEPTH	Depth of hole (algebraic sign=work direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for drilling
Q202=5 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time after every retraction in seconds
Q203=-10 ;SURFACE COORDINATE	Coordinate of the workpiece surface
Q204=20 ;2ND SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q211=0.2 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
Q395=0 ;DEPTH REFERENCE	Depth referenced to the tool tip or the cylindrical part of the tool
6 CYCL CALL	Call the cycle
7 Z+200 R0 FMAX M2	Retract the tool
8 END PGM $MDI MM	End of program

Saving NC programs from $MDI

The $MDI file is intended for short NC programs that are only needed temporarily. Nevertheless, you can store an NC program, if necessary, by proceeding as follows:

⬦	▶ Operating mode: Press the **Programming** key
PGM MGT	▶ Call the file manager: Press **PGM MGT** key.
↑	▶ Move the highlight to the **$MDI** file
COPY ABC → XYZ	▶ Copy the file: Press the **COPY** soft key
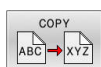
DESTINATION FILE =

▶ Enter the name under which you want to save the current contents of the $MDI file, e.g. **Hole**
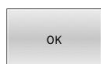
OK	▶ Press the **OK** soft key.
END	▶ To exit the file manager, press the **END** soft key

6.11 Entering miscellaneous functions M

Fundamentals

With the control's miscellaneous functions—also called M functions—you can affect:

- the program run, e.g. a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- the path behavior of the tool

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate NC block. The control displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the miscellaneous function in the programming dialog. With some miscellaneous functions, the dialog is extended so that you can enter the required parameters for this function.

In the **Manual operation** and **Electronic handwheel** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

Some miscellaneous functions take effect at the start of the NC block and others at the end, regardless of the sequence in which they were programmed.

Miscellaneous functions come into effect in the NC block in which they are called.

Some miscellaneous functions are effective block-by-block, i.e. only in the NC block in which the miscellaneous function has been programmed. When a miscellaneous function takes effect modally, you have to cancel this miscellaneous function again in a subsequent NC block (e.g., by using **M9** to switch off coolant that was switched on with **M8**). If miscellaneous functions are still active at the end of the program, the control will rescind the miscellaneous functions.

> **i** If multiple M functions were programmed in a single NC block, the execution sequence is as follows:
> - M functions taking effect at the start of the block are executed before those taking effect at the end of the block
> - If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

6.12 Miscellaneous functions for program run inspection, spindle and coolant

Overview

> ⚙ Refer to your machine manual.
>
> The machine manufacturer can influence the behavior of the miscellaneous functions described below.

M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine manufacturer)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 0 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

6.13 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/M92

Scale datum

On the scale, a reference mark indicates the position of the scale datum.
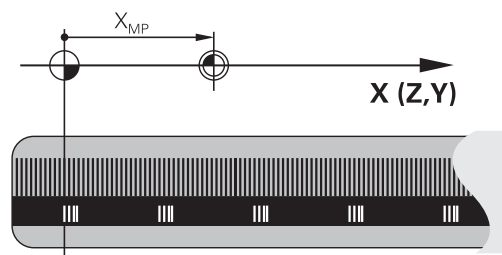
Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (such as tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine manufacturer in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: "Presetting without a 3D touch probe", Page 184

Behavior with M91 – Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, enter M91 into these NC blocks.

> ⓘ   If you program incremental coordinates in an NC block with the miscellaneous function **M91**, then these coordinates are relative to the last position programmed with **M91**. If the active NC program does not contain a position programmed with **M91**, the coordinates reference the current tool position.

The coordinate values on the control's screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF.

Further information: "Status displays", Page 68

Behavior with M92 – Additional machine reference point

> ⚙ Refer to your machine manual.
>
> In addition to the machine datum, the machine manufacturer can also define an additional machine-based position as a reference point (machine preset).
>
> For each axis, the machine manufacturer defines the distance between the machine preset and the machine datum.

If you want the coordinates in positioning blocks to be based on the machine preset, enter M92 into these NC blocks.

> ⓘ Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

M91 and M92 are effective only in the blocks in which M91 and M92 have been programmed.

M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always be referenced to the machine datum, you can inhibit presetting for one or more axes.

If presetting is inhibited for all axes, the control does not display the **SET PRESET** soft key in the **Manual operation** operating mode.

The figure shows coordinate systems with the machine and workpiece datum.
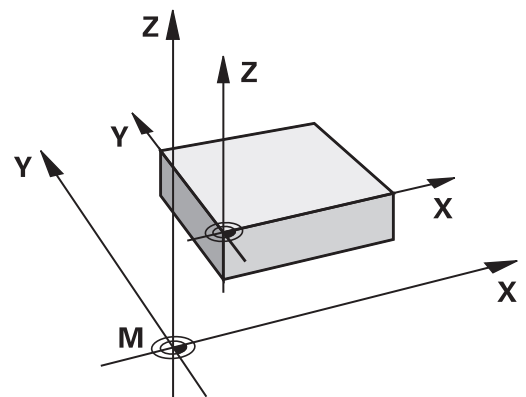
M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the defined preset.

Further information: "Showing the workpiece blank in the working space ", Page 219

7

Special Functions

7.1   Defining a counter

Application

⚙   Refer to your machine manual.
      Your machine manufacturer enables this function.

With the **FUNCTION COUNT** NC function, you control a counter
from within the NC program. This counter allows you, for example,
to define a target count up to which the control is to repeat the
NC program.

To program this behavior:

| SPEC FCT | ▶ Show the soft-key row for special functions |

| PROGRAM FUNCTIONS | ▶ Press the **PROGRAM FUNCTIONS** soft key |

| FUNCTION COUNT | ▶ Press the **FUNCTION COUNT** soft key |
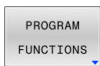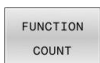
NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute
an NC program that resets the counter, any counter progress of
another NC program will be deleted.

▶ Please check prior to machining whether a counter is active.
▶ If necessary, note down the counter value and enter it again via
the MOD menu after execution.

Effect in the Test Run operating mode

You can simulate the counter in the **Test Run** operating mode. Only
the counter reading you have defined directly in the NC program is
active. The counter reading in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes

The counter reading from the MOD menu is only active in the
Program Run Single Block and **Program Run Full Sequence**
operating modes.

The counter reading remains the same after a restart of the control.

Defining FUNCTION COUNT

The **FUNCTION COUNT** NC function provides the following counter functions:

Soft key	Function
FUNCTION COUNT INC	Increase the counter by 1
FUNCTION COUNT RESET	Reset the counter
FUNCTION COUNT TARGET	Define the target count to be reached Input value: 0 to 9999
FUNCTION COUNT SET	Assign a defined value to the counter Input value: 0 to 9999
FUNCTION COUNT ADD	Increase the counter by a defined value Input value: 0 to 9999
FUNCTION COUNT REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet

Example

5 FUNCTION COUNT RESET	Reset the counter reading
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 …	Machining operation
51 FUNCTION COUNT INC	Increment the counter reading
52 FUNCTION COUNT REPEAT LBL 11	Repeat the machining operations if more parts are to be machined
53 M30	
54 END PGM	

8

MOD Functions

8.1   MOD function

The MOD functions provide additional input possibilities and
displays. In addition, you can enter code numbers to enable access
to protected areas.

Selecting MOD functions

To open the MOD menu:

| MOD | ▶ Press the **MOD** key
            > The control opens a pop-up window displaying
              the available MOD functions.
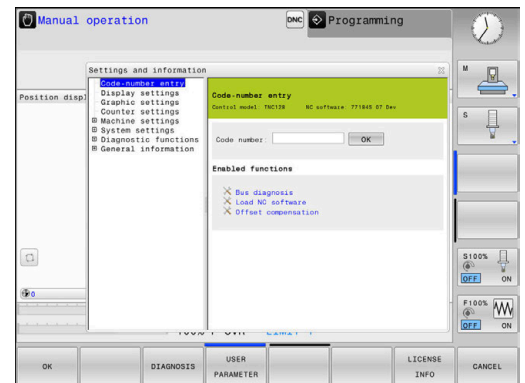
Changing the settings

There are three possibilities for changing a setting, depending on the
function selected:

- ▶ Enter a numerical value directly, e.g., when determining the
  traverse range limit
- ▶ Change the setting by pressing the **ENT** key
- ▶ Change a setting via a selection window

> ⓘ   If there are multiple possible settings available, then you
>   can show the selection box by pressing the **GOTO** key.
>   Select the desired setting with the **ENT** key. If you do not
>   wish to change the setting, close the window with the **END**
>   key.

Exiting MOD functions

To exit the MOD menu:

- ▶ Press the **CANCEL** soft key or the **END** key
- > The control closes the pop-up window.

Overview of MOD functions

The following groups with subordinate areas and functions are available independently of the selected operating mode:

Code-number entry

- Code number

Display settings

- Position display
- Measuring unit (mm/inch) for position display
- Program entry for MDI
- Show time of day
- Show the info line

Graphic settings

- Model type
- Model quality

Counter settings

- Momentary count
- Target value for counter

Machine settings

- Kinematics
- Traverse limits
- Tool-usage file
- External access
- Set up wireless handwheel

System settings

- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- TNCdiag
- Hardware configuration
- HEROS information

General Information

- Version information
- Info about machine manufacturer
- Machine information
- License information
- Machine times
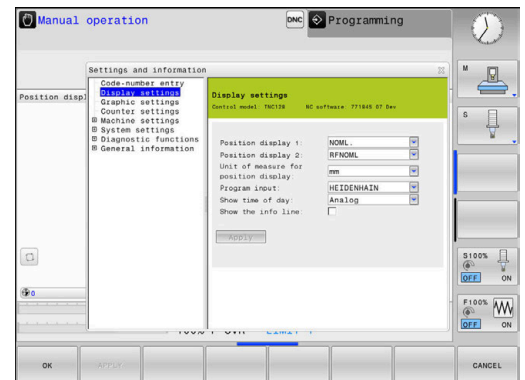
> **ℹ** The **Info about machine manufacturer** area is available after the machine manufacturer has defined the machine parameter **CfgOemInfo** (no. 131700).
>
> The **Machine information** area is available after the machine operator has defined the machine parameter **CfgMachineInfo** (no. 131600).

8.2　Displaying software numbers

Application

In the **Version Information** MOD area of the **General Information** group, the control displays the following information on the software:

- **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-SW**: Number or name of the PLC software (managed by the machine manufacturer)

8.3 Entering the code number

Application

The control requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configuration of Ethernet interface	NET123
Enabling of special functions during Q-parameter programming	555343
Resetting active code numbers	0

The control shows in the code number dialog whether CAPS LOCK is active.

Functions provided to the machine manufacturer by the code number dialog

The two soft keys **OFFSET ADJUST** and **UPDATE DATA** are shown in the MOD menu of the control.

The **OFFSET ADJUST** soft key enables automatic determination and subsequent saving of an offset voltage required for analog axes.

⚙️ Refer to your machine manual.
This function may only be used by trained personnel!

The **UPDATE DATA** soft key enables the machine manufacturer to install software updates on the control.

NOTICE

Caution: Data may be lost!

Not following the instructions when installing updates can cause a loss of data.

▶ Always follow the instructions when installing software updates!

▶ Refer to your machine manual.

8.4   Loading the machine configuration

Application

> **NOTICE**
>
> **Caution: Data may be lost!**
>
> The **RESTORE** function irrevocably overwrites the current machine configuration with the backup files. The control does not perform an automatic backup before the **RESTORE** function is executed. The current files are thus permanently lost.
>
> ▶ Perform a backup of the current machine configuration prior to the **RESTORE** function
>
> ▶ Use the function only in consultation with the machine manufacturer

Your machine manufacturer can provide you with a backup including a machine configuration. After entering the **RESTORE** keyword, you can restore the backup on your machine or programming station.

To restore a backup:

▶ Select the **Code-number entry** MOD function

▶ Enter the **RESTORE** keyword

▶ Press the **OK** soft key

▶ Select the backup file in the control's file manager (e.g., BKUP-2013-12-12_.zip)

> The control opens the pop-up window for the backup.

▶ Press Emergency Stop

▶ Press the **OK** soft key

> The control starts restoring the backup.

8.5 Selecting the position display

Application

In the **Display settings** group, you can influence the display of the coordinates for **Manual operation** mode as well as for the **Program run, full sequence** and **Program run, single block** operating modes:

The figure on the right shows the different tool positions:

- Initial position
- Target position of the tool
- Workpiece datum
- Machine datum

You can select the following coordinates for the control's position displays:

Display	Function
NOML	Nominal position: The value currently commanded by the control
	ⓘ The NOML and ACTL displays differ solely with regard to following error.
ACTL	Actual position; current tool position
	⚙ Refer to your machine manual. Your machine manufacturer defines whether the ACTL and NOML display deviates from the programmed position by the DL oversize of the tool call.
REF ACTL	Reference position; actual position relative to the machine datum
REF NOML	Reference position; nominal position relative to the machine datum
LAG	Servo lag; difference between nominal and actual positions
ACTDST	Distance remaining to the programmed position in the input coordinate system; difference between actual and target positions Examples with Cycle **11**: ▶ Scaling factor 0.2 ▶ L IX+10 > The ACTDST display shows 10 mm. > The scaling factor does not have any influence.

Display	Function
REFDST	Distance remaining to the programmed position in the machine coordinate system; difference between actual and target positions Examples with Cycle **11**: ▶ Scaling factor 0.2 ▶ L IX+10 > The REFDST display shows 2 mm. > The scaling factor has an effect on the distance and thus on the display.
M118	Traverse paths that were executed with handwheel superimpositioning function (**M118**)

With the **Position display 1** MOD function, you can select the position display in the status display.

With the **Position display 2** MOD function, you can select the position display in the additional status display.

8.6 Setting the unit of measure

Application

Use the **Unit of measure for position display** MOD function in the **Display settings** group to set the coordinate display to mm or inches.

- 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.

8.7 Graphic settings

Use the **Simulation parameters** MOD function in the **Graphic settings** group to select the model type and model quality for the **Test Run** operating mode.

To define the graphic settings:

▶ Select the **Graphic settings** group in the MOD menu

▶ Select **Model type**

▶ Select **Model quality**

▶ Press the **APPLY** soft key

▶ Press the **OK** soft key.

> The control saves the selected settings.
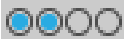
In the **Test Run** operating mode, the control displays the icons of the active graphic settings.

In the **Simulation parameters** MOD function, the following settings are available:

Model type

Icon	Choice	Properties	Application
	3D	Very true to detail, heavy time and processor consumption	Milling with undercuts,
	2.5D	Fast	Milling without undercuts
	No model	Very fast	Line graphics

Model quality

Icon	Choice	Properties
	Very high	■ High data transfer rate ■ Accurate display of the tool geometry ■ Block end points and block numbers can be displayed
	High	■ High data transfer rate ■ Accurate display of the tool geometry
	Medium	■ Medium data transfer rate ■ Approximation of the tool geometry
	Low	■ Low data transfer rate ■ Less accurate approximation of the tool geometry

Aspects to be observed with the Graphic settings

Besides the MOD settings, the NC program has a great influence on the simulation result. Setting the highest model quality with many very short NC blocks might slow down the simulation.

Setting a low model quality, in turn, may lead to a distorted simulation result because very short NC blocks become invisible due to the low resolution.

HEIDENHAIN recommends the following settings:

- Fast visualization of a 3-axis program or test of an NC program for feasibility
  - **Model type**: 2.5D
  - **Model quality**: medium
- Test of the NC program by means of the graphic simulation
  - **Model type**: 3D
  - **Model quality**: very high

8.8   Setting the counter

The **Counter** MOD function in the **Counter settings** group allows you to change the current counter reading (actual value) and the target value (nominal value).

To define the counter:

▶ Select the **Counter settings** group in the MOD menu
▶ Define the **Momentary count**
▶ Define the **PGM for counter**
▶ Press the **APPLY** soft key
▶ Press the **OK** soft key
> The control saves the selected settings.

The control applies the defined values to the status display.

The following soft keys are available in the **Counter** MOD functions:

Soft key	Function
RESET	Reset count
+	Increase count
-	Lower count

You can also enter the values directly with a connected mouse.

Further information: "Defining a counter", Page 260

8.9 Changing the machine settings

Selecting the kinematics

> ⚙ Refer to your machine manual.
> The **Kinematics** function will be configured and enabled by your machine manufacturer.

NOTICE

Danger of collision!

All stored kinematics can also be selected as active machine kinematics. By this means, all manual movements and machining operations are executed using the selected kinematics. All subsequent axis movements pose a risk of collision!

▶ Use the **Kinematics** function exclusively in the **Test Run** operating mode

▶ Use the **Kinematics** function for selecting the active machine kinematics only if required

---

The **Kinematics** MOD function in the **Machine settings** group allows you to select different kinematics as machine kinematics for the test run. You can use this function to test NC programs whose kinematics does not match the active machine kinematics.

The various kinematics must be defined and enabled by the machine manufacturer. When you select a kinematics for the test run, the machine kinematics remains unaffected.

To change the kinematics:

▶ Select the **Machine settings** group in the MOD menu

▶ Select the **Kinematics** function

▶ Select the desired kinematics for the **SIM** channel

▶ Press the **APPLY** soft key

▶ Press the **OK** soft key

> The control saves the selected kinematics for the **Test Run** operating mode.

> ℹ Ensure that you have selected the correct kinematics in the Test Run operating mode for checking your workpiece.

Entering traverse limits

⚙ Refer to your machine manual.
Your machine manufacturer configures and enables the
Traverse limits function.

The **Traverse limits** MOD function in the **Machine settings** group
enables you to limit the actually usable traverse path within the
maximum range of traverse. Thus, you can define traverse limits on
each axis (e.g., to protect an indexing head from collision).

To define traverse limits:

▶ Select the **Machine settings** group in the MOD menu
▶ Select the **Traverse limits** function
▶ Define a value in the **Lower limit** or **Upper limit** column
  or
▶ Apply the current position by pressing the
  ACTUAL POSITION CAPTURE soft key
▶ Press the **APPLY** soft key
> The control validates the defined values.
▶ Press the **OK** soft key
> The control saves the defined traverse limits.

ℹ Operating notes:
  ■ The protection zone becomes active automatically as
    soon as you have set a valid traverse limit in an axis.
    The settings are kept even after the control has been
    restarted.
  ■ The only way to deactivate the traverse limits is to
    delete all values or to press the **EMPTY EVERYTHING**
    soft key.
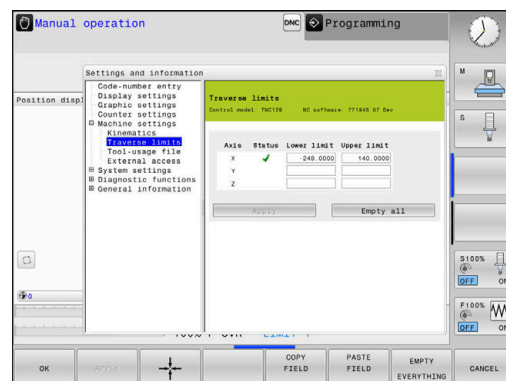
Software limit switch with modulo axes

If you set software limit switches for modulo axes, make sure to comply with the following basic conditions:

■ The lower limit is greater than −360° and less than +360°
■ The upper limit is not negative and less than +360°
■ The lower limit is not greater than the upper limit
■ The difference between the upper and the lower limit is less than 360°

If these conditions are not met, you will not be able to move the modulo axis. The TNC 128 displays an error message.

If modulo limit switches are active, a movement is allowed if the target position or an equivalent position is within the allowed range. Equivalent positions are positions that differ from the target positions by an offset of n · 360° (where n is any integer). The direction of motion is derived automatically because there is always only one equivalent position that the axis can move to—with the exception described below.

Example:
For the modulo axis C, the limit switches have been set to −80° and +80°. The axis is at 0°. If you program **L C+320** now, the C axis will move to −40°.

If an axis is positioned outside the limit switch range, it can only be moved towards the nearer limit switch.

Example:
The limit switches have been set to −90° and +90°. The C axis is at −100°.
In this case, the C axis must move in the positive direction with its next movement so that moving by **L C+I5** works, while **L C-I5** would result in a limit switch violation.

Exception:
The axis is positioned exactly in the middle of the forbidden range. Thus, the distance to both limit switches is the same. In this case, the movement can go to either direction. This results in the special situation that the axis can move to two equivalent positions if the target position is within the allowed range. In this case, the axis will be moved to the nearer equivalent position, i.e. the shorter path is used. If both equivalent positions are equidistant (i.e., 180° away), the control selects the direction of motion according to the programmed value.

Example:
The limit switches have been set to C−90°, C+90° and the axis is at 180°.
If you program **L C+0**, the C axis will move to 0. The same is true when programming **L C-360**, etc. However, if you program **L C+360** (or L C+720, etc.), then the C axis will move to 360°.

Generating a tool usage file

⚙ Refer to your machine manual.
The tool-usage-test function must be enabled by your machine manufacturer.

Using the **Tool-usage file** MOD function in the **Machine settings** group, you can define whether the control never, once, or always creates a tool usage file. The settings for the test run and the program run operating modes must be defined separately.

To change the settings in the tool usage file:

▶ Select the **Machine settings** group in the MOD menu

▶ Select the **Tool-usage file** function

▶ Select the setting for **Program Run, Full Sequence/Single Block**

▶ Select the setting for **Test Run**

▶ Press the **APPLY** soft key

▶ Press the **OK** soft key

> The control saves the defined settings.

Permitting or restricting external access

⚙ Refer to your machine manual.
The machine manufacturer can configure the external access options.

The **External access** MOD function in the **Machine settings** group allows you to permit or block access to the control. After blocking external access, you can no longer connect with the control. In that case, it is also no longer possible to exchange data through a network or a serial connection,(e.g., with the **TNCremo** software).

To block external access:

▶ Select the **Machine settings** group in the MOD menu

▶ Select the **External access** function

▶ Set the **EXTERNAL ACCESS ON/OFF** soft key to **OFF**

▶ Press the **OK** soft key

> The control saves the settings.

ⓘ Once the control is accessed externally, the following icon appears:

⇄
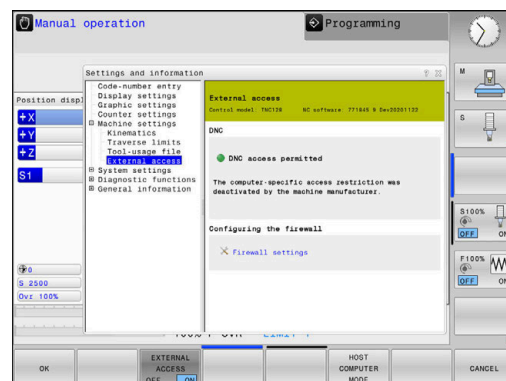
Computer-specific access control

If your machine manufacturer has set up computer-specific access control (machine parameter **CfgAccessControl** no. 123400), you can permit access for up to 32 connections authorized by you.

To create a new connection:

▶ Press the **ADD NEW** soft key

> The control opens an input window for you to enter the connection data.

Access settings

Host name	Host name of the external computer
Host IP	Network address of the external computer
Description	Additional information
	The text is shown in the overview list.

Type:

Ethernet	Network connection
Com 1	Serial interface 1
COM 2	Serial interface 2

Access rights:

Inquire	For external access, the control opens a query dialog
Deny	Do not permit network access
Permit	Permit network access without query

> **i** 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 authentication.
>
> If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

If you assign the **Inquire** access right to a connection, and if access is gained from this address, then the control opens a pop-up window.
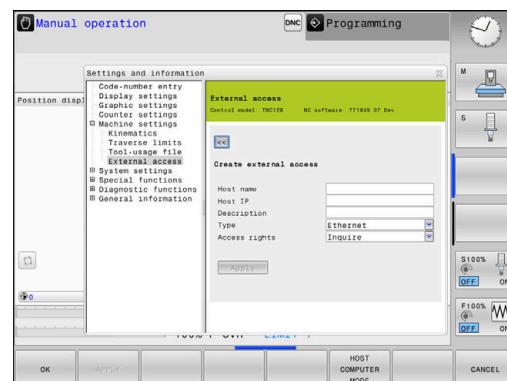
You must decide if you permit or deny external access in the dialog:

External access	Permission
Yes	Permit once
Always	Permit continuously
Never	Deny continuously
No	Deny once

> 🛈   In the overview list, an active connection is shown with a green symbol.

Host computer operation

> ⚙   Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

With the **HOST COMPUTER MODE** soft key you can transfer the command to an external host computer (e.g., in order to run NC programs).

The requirements to be met for starting host computer mode include the following:

- Dialogs, such as **GOTO** or **Block scan**, are closed
- No program run is active
- Handwheel is inactive

To start the host computer mode:

▶ Select the **Machine settings** group in the MOD menu
▶ Select the **External access** function
▶ Press the **HOST COMPUTER MODE** soft key
> The control shows an empty screen with the **Host computer is active** pop-up window.

> ⚙   Your machine manufacturer can specify that the host computer mode can automatically be activated externally.

To exit the host computer mode:

▶ Press the **HOST COMPUTER MODE** soft key again
> The control closes the pop-up window.

Allowing a secure connection

The **KEY MANAGEMENT** soft key opens the **Certificate and keys** window. This window can be used to define settings for secure connections via SSH.

Further information: "User authentication from external applications", Page 356

8.10 Configuring the HR 550 FS radio handwheel

Application

ℹ️ This setup dialog is managed by the HEROS operating system.

After you have changed the conversational language on your control, you need to restart the control to enable the new language.

Use the **SET UP WIRELESS HANDWHEEL** soft key to configure the HR 550 FS wireless handwheel. The following functions are available:

- Assigning the handwheel to a specific handwheel holder
- Setting the transmission channel
- Analyzing the frequency spectrum for determining the optimum transmission channel
- Selecting transmitter power
- Statistical information on the transmission quality

ℹ️ Any changes or modifications that were not expressly approved by the party responsible for conformity could void your authority to operate the device.

This device complies with Part 15 of the FCC Rules and the RSS standard(s) of Industry Canada for license-exempt equipment.

Operation is subject to the following conditions:
1 The device must not cause any harmful interference.
2 The device must be tolerant to any interferences it receives, including interference that might impair its operation.

Assigning the handwheel to a specific handwheel holder

To assign a handwheel to a specific handwheel holder, the handwheel holder must be connected to the control hardware.

To assign a handwheel to a specific handwheel holder:
▶ Place the handwheel in the handwheel holder
▶ Select the **Machine settings** group in the MOD menu
▶ Press the **SET UP WIRELESS HANDWHEEL** soft key
> The control opens a pop-up 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.
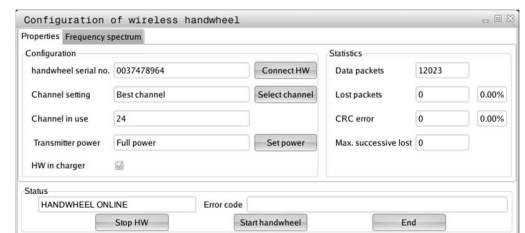
Setting the transmission channel

If the wireless handwheel is started automatically, then the control tries to select the transmission channel that provides the best transmission signal.

To manually set the radio channel:

▶ Select the **Machine settings** group in the MOD menu
▶ Press the **SET UP WIRELESS HANDWHEEL** soft key
> The control opens a pop-up 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 (smallest bar)
▶ 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.
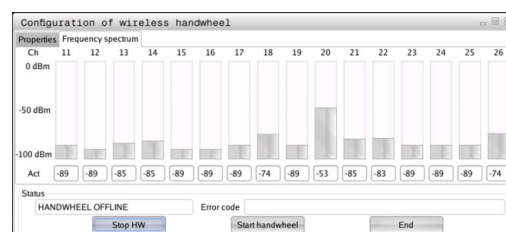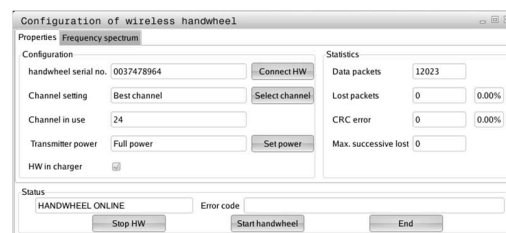
Selecting the transmitter power

> ⓘ   A reduction in transmission power decreases the range of the wireless handwheel.

To set the transmitter power of the handwheel:

▶ Select the **Machine settings** group in the MOD menu
▶ Press the **SET UP WIRELESS HANDWHEEL** soft key
> The control opens a pop-up window.
▶ Select the **Set power** button
> The control displays the three available power settings.
▶ Select the desired transmitter power setting
▶ Select the **END** button
> The control saves the configuration.
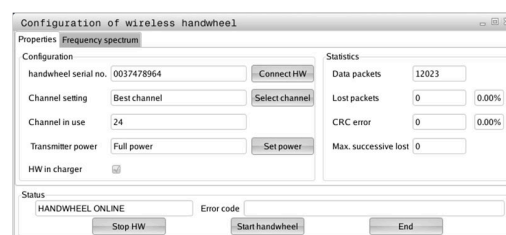
Statistics

To display the statistical data:

- ▶ Select the **Machine settings** group in the MOD menu
- ▶ Press the **SET UP WIRELESS HANDWHEEL** soft key
- > The control opens a pop-up window.

Under **Statistics**, 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 transmission channel", Page 282

Further information: "Selecting the transmitter power", Page 282
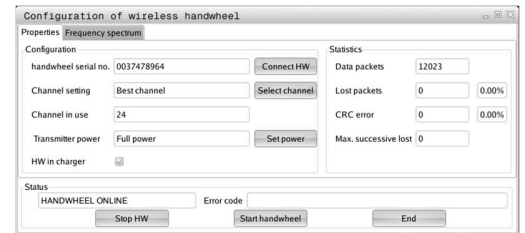
8.11 Changing the system settings

Set the system time

With the **Set the system time** MOD function in the **System settings** group, you can set the time zone, date, and time manually or by means of an NTP server synchronization.

To set the system time manually:

▶ Select the **System settings** group in the MOD menu

▶ Press the **SET DATE/ TIME** soft key

> The control opens a pop-up window.

▶ In the **Time zone** area, select the desired time zone

▶ Press the **NTP off** soft key, if necessary

> The control enables the **Set the time manually** check box.

▶ Change the date and time, if required

▶ Press the **OK** soft key

> The control saves the settings.

To set the system time using a NTP server:

▶ Select the **System settings** group in the MOD menu

▶ Press the **SET DATE/ TIME** soft key

> The control opens a pop-up window.

▶ In the **Time zone** area, select the desired time zone

▶ Press the **NTP on** soft key, if necessary

> The control enables the **Synchronize the time over NTP server** check box.

▶ Enter the host name or the URL of an NTP server

▶ Press the **Add** soft key

▶ Press the **OK** soft key

> The control saves the settings.

8.12 Diagnostic functions

Bus diagnosis

⚙ This function is protected by a code number.

Only use this function after consultation with your machine manufacturer.

The **Diagnostic functions** group of the **Bus diagnosis** MOD area allows the machine manufacturer to retrieve the bus system data.

Hardware configuration

⚙ Only use this function after consultation with your machine manufacturer.

In the **Diagnostic functions** group of the **Hardware configuration** MOD area, the control displays the nominal and actual hardware configurations in the **HwViewer**.

If the control detects a hardware change, it automatically opens the error window. The soft keys displayed there allow you to open the **HwViewer**.

The changed hardware component is color-highlighted.

HeROS information

In the **Diagnostic functions** group of the **HeROS information** MOD area, the control displays details on the operating system.

Besides the information on the control type and the software version, this MOD area also shows the current CPU and memory usage values.

8.13 Displaying operating times

Application

In the **General Information** group of the **MACHINE TIMES** MOD area, the control displays the following operating times:

Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service

⚙ Refer to your machine manual.

The machine manufacturer can provide further operating time displays.
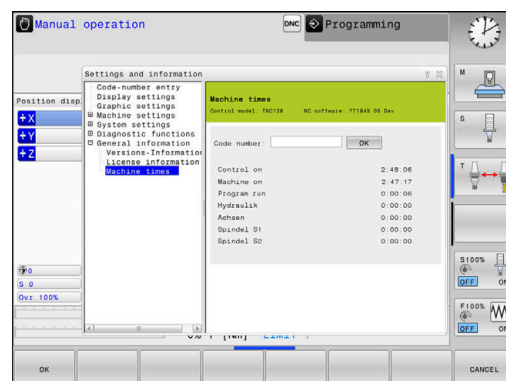
9

HEROS Functions

9.1 Window manager

> ⚙️ Refer to your machine manual.
> The machine manufacturer determines the scope of function and behavior of the window manager.

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

> ℹ️ 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.

Overview of taskbar

In the taskbar, you can choose different workspaces by mouse click.

The control provides the following workspaces:

- Workspace 1: Active machine operating mode
- Workspace 2: Active programming operating mode
- Workspace 3: applications of the machine manufacturer (optionally available)
- Workspace 4: applications of the machine manufacturer (optionally available)

You can also select other applications from the taskbar which you have started in addition to the control software, e.g. **TNCguide**.
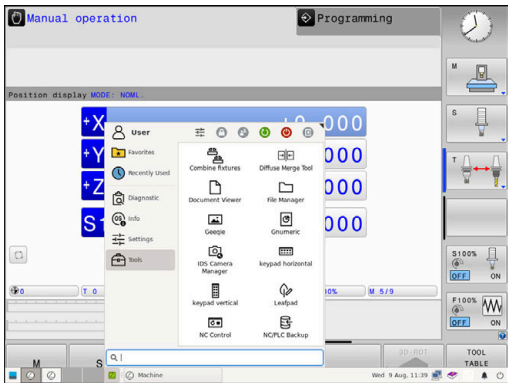
> **i** All open applications to the right of the green HEIDENHAIN icon can be moved between the workspaces any way you desire by pressing and holding the left mouse button.

Click the green HEIDENHAIN icon to open a menu in which you can get information, make settings or start applications.

Area	Function
Header	■ User name **Further information:** "Current User", Page 367 ■ 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:** "Switching or logging out users", Page 363
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 292 ■ **Portscan OEM**: Available only to authorized specialists ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically > ⚙ If you want to use **TNCdiag**, contact your machine manufacturer. ■ **TNCscope** Software for data recording

Area	Function
Settings	■ **Adjust screen brightness**: Adjust screen brightness
	■ **Screensaver**: Configure the screensaver
	Further information: "Screensaver with lockout", Page 363
	■ **Current User**
	Further information: "Current User", Page 367
	■ **Date/Time**: Set date and time
	■ **Firewall**: Configure the firewall
	Further information: "Firewall", Page 302
	■ **HePacketManager**: Available only to authorized specialists
	■ **HePacketManager Custom**: Available only to authorized specialists
	■ **Language/Keyboards**: Select system dialog language and keyboard version—the control overwrites the user interface language setting when starting with the language setting of the machine parameter **CfgDisplayLanguage** (no. 101300)
	■ **Network**: Define network settings
	Further information: "Ethernet interface ", Page 312
	■ **OEM Function Users**: Edit machine manufacturer function user
	Further information: "HEIDENHAIN function users", Page 350
	■ **Printer**: Configure and manage printer
	Further information: "Printer", Page 294
	■ **Screenshot Config**
	In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).
	■ **SELinux**: Define safety software for Linux-based operating systems
	Further information: "SELinux security software", Page 332
	■ **Shares**: Connect and manage external network drives
	Further information: "Settings for network drives", Page 324
	■ **UserAdmin**: Configure user administration
	Further information: "Configuring user administration", Page 334
	■ **VNC**: Define the setting for external software accessing the control, e.g., for maintenance procedures (Virtual Network Computing)
	Further information: "VNC", Page 297
	■ **WindowManagerConfig**: Available only to authorized specialists
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 the control **Further information:** "Switching or logging out users", Page 363 ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Diffuse Merge Tool**: Compare and merge text files ■ **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**: Create backup file **Further information:** "Backup and restore", Page 300 ■ **NC/PLC Restore**: Restore backup file **Further information:** "Backup and restore", Page 300 ■ **QupZilla**: Alternative web browser for touch operation ■ **Real VNC Viewer**: Define the setting for external software accessing the control (e.g., for maintenance purposes (Virtual Network Computing)) ■ **Ristretto**: Open graphics ■ **Secure Remote Access** **Further information:** "Secure Remote Access", Page 293 ■ **TNCguide**: Call the help system ■ **TouchKeyboard**: Open keyboard for touch operation ■ **Web Browser**: Start the web browser ■ **Xarchiver**: Extract or compress directories
Searching	Full-text search of individual functions

> ⓘ The applications available under **Tools** can be started directly by selecting the corresponding file type in the control's file manager.
> **Further information:** "Software tools for management of external file types", Page 89

Portscan

The PortScan function allows the cyclic or manual searching for all open, incoming TCP and UDP listen ports on the system. All ports found are compared with whitelists. If the control finds a non-listed port it shows a corresponding pop-up window.

The **HEROS menu Diagnostic** menu contains the **Portscan** and **Portscan OEM** applications for this purpose. **Portscan OEM** is only executable after entering the machine manufacturer password.

The **Portscan** function searches for all open, incoming TCP and UDP ports on the system and compares them to four whitelists stored in the system:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports for machine manufacturer-specific functions (e.g., for Python applications, external applications): **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

For each entry, the whitelist specifies the type of port (TCP/UDP), the port number, the providing program, and optional comments. If the automatic port scan function is active, only ports listed in the whitelists can be open. Non-listed ports trigger a notification window.

The result of the scan is saved to a log file (LOG:/portscan/scanlog and LOG:/portscan/scanlogevil) and, if new ports are found that are not listed in one of the whitelists, these are displayed.

Starting Portscan manually

To manually start **Portscan**:

- ▶ Open the task bar at the bottom edge of the screen
  Further information: "Window manager", Page 288
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- > The control opens the **HeRos Portscan** pop-up window.
- ▶ Press the **Start** button

Cyclically starting Portscan

To automatically start the Portscan cyclically:

- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- > The control opens the **HeRos Portscan** pop-up window.
- ▶ Press the **Automatic update on** button
- ▶ Set the time interval with the slider

Secure Remote Access

Application

Secure Remote Access SRA allows you to set up an encrypted connection between a PC and your control via the Internet. SRA allows the control to be displayed and operated on a PC, such as for service trainings or remote maintenance.

Related topics

- VNC settings

  Further information: "VNC", Page 297

Requirements

- Existing Internet connection

  Further information: "Network configuration with Advanced Network Configuration", Page 318

- The following settings in the **VNC settings** window:

  - **Enable RemoteAccess and IPC** check box is active
  - In the **Enabling other VNC** area, the **Inquire** or **Permitted** check box is active

  Further information: "VNC", Page 297

- PC with paid RemoteAccess software including the extension **Secure Remote Access**

  HEIDENHAIN website

  > 📖 For more information, refer to the integrated help system of RemoteAccess.
  >
  > You can open the context-sensitive help function of the RemoteAccess software by pressing the **F1** key.

Description of function

To navigate to this function:

Tools ▶ Secure Remote Access

The PC provides a ten-digit session ID for you to enter in the
HEIDENHAIN Secure Remote Access window.

SRA enables connection via an VPN server.

In the **Extended** area, the control shows the progress of the
connection setup.

The **HEIDENHAIN Secure Remote Access** window provides the
following buttons:

Button	Function
Connect	The control starts the connection with the entered session ID.
Update	The control manually searches for updates for SRA.
	The control automatically searches for available updates when you open the **HEIDENHAIN Secure Remote Access** window.
	If an update is available, you can install it. The control restarts during the update.
Config.	The control opens the **Network settings** window.
	Only for network specialists
Show log	The control opens the log files of the SRA.

Notes

If, in the **VNC settings** window, you set the **Enabling other VNC**
setting to **Inquire**, you can permit or deny any connection.

Printer

The **Printer** function in the **HEROS menu** enables you to add and
manage printers.

Displaying the printer settings

To open the printer settings:

▶ Open the taskbar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Printer** menu item

> The control opens the **Heros Printer Manager** dialog.
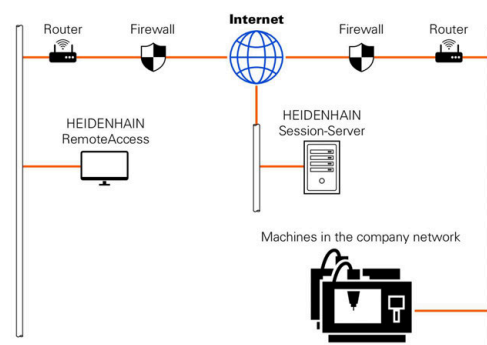
Overview of the soft keys

Soft key	Function	Meaning
CREATE	**CREATE**	Creates a printer
CHANGE	**CHANGE**	Adapts the properties of the selected printer
COPY	**COPY**	Creates a copy of the selected printer
		Initially, the copy has the same properties as the copied printer. This can be useful for printing both portrait and landscape formats on the same printer
DELETE	**DELETE**	Deletes the selected printer
UP	**UP**	Selects a printer
DOWN	**DOWN**	
STATUS	**STATUS**	Displays the status information of the selected printer
PRINT A TEST PAGE	**PRINT A TEST PAGE**	Prints a test page on the selected printer

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 the **CHANGE** soft key

> The control opens the **Change the printer** dialog.

For each printer, the following properties can be set:

Setting	Meaning
Name of the printer	Customize the printer name
Connection	Select the connection
	■ **USB** – Assign the USB connection here. The control automatically displays the name.
	■ **Network** – Enter the network name or the IP address of the printer here. In addition, specify the port of the network printer here (default: 9100)
	■ **Printer %1 not connected**
Timeout	Delays the printing process
	The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**.
	Use this setting if the file to be printed is populated with FN functions (e.g., when probing).
Standard printer	Select the default printer
	When creating the first printer, this setting is automatically entered.
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

Requirements for the connected printer

> ⓘ The connected printer must be PostScript-enabled.

The control can communicate only with printers that understand PostScript emulation such as KPDL3. With some printers, the PostScript emulation can be set in the printer menu.

To change the printer settings:

▶ Select the printer menu

▶ Select printing settings

▶ Select KPDL for the emulation

▶ Confirm if required

Print

Print options:

- Copying the file to be printed into the **PRINTER** drive:

  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job is complete.

  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

Printable files:

- Text files
- Graphic files
- PDF files

VNC

Use the **VNC** function to configure the behavior of the various VNC clients. This includes, for example, operation via soft keys, mouse, and the alphabetic keyboard.

The control provides the following options:

- List of permitted clients (IP address or name)
- Password for the connection
- Additional server options
- Additional settings for assigning the focus

> ⚙ Refer to your machine manual.
>
> For multiple clients or operating units, the focus assignment procedure depends on the design and the operating situation of the machine.
>
> This function must be adapted by your machine manufacturer.
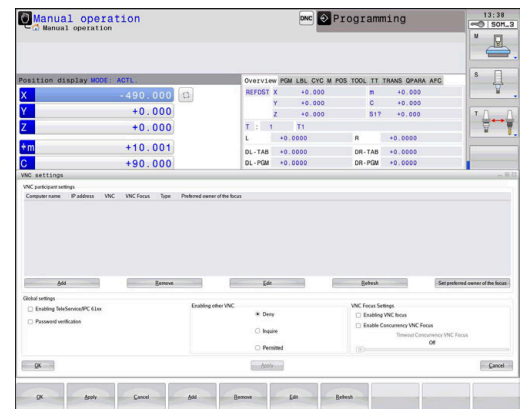
Opening the VNC settings

To open the **VNC settings**:

▶ Open the task bar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **VNC** menu item

> The control opens the **VNC settings** pop-up window.

The control provides the following options:

■ Add: Add new VNC viewer/client

■ Remove: Deletes the selected client. Only possible with manually entered clients.

■ Edit: Edit the configuration of the selected client

■ Update: Updates the display. Required with connection attempts during which the dialog is open.

VNC settings

Dialog	Option	Meaning
VNC participant settings	**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 TeleService connection ■ DHCP Other computer that obtains an IP address from this computer
Firewall warning		Warnings and information about if the VNC protocol has not been authorized for all VNC clients due to firewall settings on the control. **Further information:** "Firewall", Page 302.
Global settings	**Enable RemoteAccess and IPC**	Connection is always permitted
	Password verification	The client must enter a password for verification. If this option is active, the password must be entered when the connection is made.

Dialog	Option	Meaning
Enabling other VNC	**Deny**	Access generally denied to all other VNC clients.
	Inquire	During connection attempts a corresponding dialog is opened.
	Permitted	Access is generally granted to all other VNC clients.
VNC Focus Settings	**Enabling VNC focus**	Enable focus assignment for this system. Otherwise there is no central focus assignment. In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus.
	Enable Concurrency VNC Focus	In the default setting, the focus is actively reassigned by the owner of the focus by clicking the focus symbol. This means that the owner of the focus must first release the focus by clicking the focus symbol before any other client can retrieve the focus. If concurrency focus is selected, any client can retrieve the focus at any time without having to wait for the current owner of the focus to release it.
	Timeout Concurrency VNC Focus	Time period within which the current owner of the focus can object to the focus being withdrawn or can prevent the reassignment of the focus. If a client requests the focus, a dialog in which the reassignment of focus can be refused appears on all clients' screens.
Focus symbol	⬛⇒🖳	Current status of VNC focus on the respective client: Focus is owned by other client. Mouse and alphabetic keyboard are disabled.
	✓⇒🖳	Current status of VNC focus on respective client: Focus is owned by current client. Entries can be made.
	🖳⇒?🖳	Current status of VNC focus on the respective client: Request by the owner of the focus to give the focus to another client. Mouse and alphabetic keyboard are disabled until the focus is assigned unambiguously.

If **Enable Concurrency VNC Focus** is selected, a pop-up window appears. This dialog allows you to deny the focus change to the requesting client. If you fail to do so, the focus changes to the requesting client after the set time limit.

> ℹ️ Enable the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose (e.g., ITC industrial computers).

Backup and restore

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 locally, on a network drive, or to USB storage devices.

The backup program generates a ***. tncbck** file that can also be processed by the PC tool TNCbackup (part of TNCremo). The Restore program can restore these files as well as those from existing TNCbackup programs. If you select a *. tncbck file in the File Manager of the control, the **NC/PLC Restore** program will be launched automatically.

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** soft keys. Specific actions for steps are selectively displayed as soft keys.

Open NC/PLC Backup or NC/PLC Restore

To open the function:
- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the **HEROS menu**
- ▶ Select the **Tools** menu item
- ▶ Open the **NC/PLC Backup** or **NC/PLC Restore** menu item
- > The control opens the pop-up window.

Backing up data

To back-up the control data:
- ▶ Select **NC/PLC Backup**
- ▶ Select type
  - ◾ Back up the **TNC:** drive
  - ◾ Back up the directory tree: Select the directory for backup in the file management
  - ◾ Backup machine configuration (for machine manufacturer only)
  - ◾ Complete backup (for machine manufacturer only)
  - ◾ Comment: Freely configurable comment for the backup
- ▶ Select the next step with the **FORWARD** soft key
- ▶ Stop the control if required with the **STOP NC SOFTWARE** soft key
- ▶ Define the exclusion rules
  - ◾ User preset rules
  - ◾ Write own rules to the table
- ▶ Select the next step with the **FORWARD** soft key
- > The control generates a list of files for backing up.
- ▶ Check the list. Deselect files if necessary.
- ▶ Select the next step with the **FORWARD** soft key
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select the next step with the **FORWARD** soft key
- > The control generates the backup file.
- ▶ Confirm with the **OK** soft key
- > The control concludes the backup process and restarts the NC software.

Restore 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 **NC/PLC Restore**
▶ Select the archive to be restored
▶ Select the next step with the **FORWARD** soft key
> The control generates a list of files for restoring.
▶ Check the list. Deselect files if necessary.
▶ Select the next step with the **FORWARD** soft key
▶ Stop the control if required with the **STOP NC SOFTWARE** soft key
▶ Extract the archive
> The control restores the files.
▶ Confirm with the **OK** soft key
> The control restarts the NC software.

9.2   Firewall

Application

You have the possibility to set up a firewall for the primary network interface of the control and a sandbox, if needed. You can configure it so that incoming network traffic is blocked and/or a message is displayed, depending on the sender and the service.

The firewall cannot be started for the secondary network interface of the control (i.e., the machine network).

If you activate the firewall, the control displays a corresponding icon at the bottom right in the taskbar. The symbol changes depending on the safety level that the firewall was activated with, and informs about the level of the safety settings:

Icon	Meaning
	No firewall protection yet although it has been activated according to the configuration. This can happen, for example, if a dynamic IP address is used in the network interface configuration, but the DHCP server has not yet assigned such an address.
	Firewall active with medium security level
	Firewall is activated with a high security level. All services except for SSH are blocked.

> **ℹ** Have your network specialist check and, if necessary, change the standard settings.

Configuring the firewall

To configure the firewall:

▶ Open the task bar at the bottom edge of the screen

▶ Press the green HEIDENHAIN button to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Firewall** menu item

> The control opens the **Firewall settings** dialog.

Activate the firewall with the pre-set default settings:

▶ Set the **active** option for switching on the firewall

▶ Press the **Set standard values** button

▶ Confirm the changes with the **Apply** function

> The control activates the default settings recommended by HEIDENHAIN.

▶ Exit the dialog with the **OK** function

Firewall settings

Setting	Meaning
Active	Switch the firewall on or off

Setting	Meaning
Interface	Select the interface
	Selection of the **eth0** interface usually corresponds to X26 of the MC main computer. **eth1** corresponds to X116.
	You can check the interfaces in the network settings on the Interfaces tab. For main computer units with two Ethernet interfaces, the DHCP server for the machine network is active for the second, non-primary, interface by default. With this setting, the firewall cannot be activated 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 option is set, the control no longer responds to a PING request
Service	This column contains the short names of the services that are configured with this dialog. For the configuration, it is not important here whether the services themselves have been started.

- **DNC** designates the service the DNC server provides via the RPC protocol for external applications that were developed with the RemoTools SDK (port 19003)

> For more detailed information, consult the RemoTools SDK manual.

- **LDAPS** includes the server on which the user data and the user administration configuration are saved.
- **LSV2** includes the functionality for **TNCremo**, Teleservice, and other HEIDENHAIN PC Tools (port 19000)

> 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 authentication.
>
> If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

- **SMB** only refers to incoming SMB connections (i.e., if a Windows share is created on the NC). Outgoing SMB connections (i.e., if a Windows share is connected to the NC) are not affected by this setting.
- **SSH** stands for the Secure Shell protocol (port 22). As of HEROS 504, LSV2 can be executed securely via this SSH protocol while user administration is active.
  Further information: "User authentication from external applications", Page 356
- **VNC** protocol means access to the screen contents. If you block this service, the screen content can no longer be accessed, not even with the Teleservice programs from HEIDENHAIN (e.g., to create a **Screenshot**) In this case, the HEROS VNC configuration dialog shows a warning that **VNC** is disabled in the firewall.

Setting	Meaning
Method	Configure accessibility
	Under **Method**, you can configure whether the service is accessible:
	■ **Prohibit all**, cannot be accessed by anyone
	■ **Permit all**, can be accessed by everyone
	■ **Permit some**, can be accessed only by individual clients
	If you set **Permit some**, you must also specify the computer that you wish to grant access to the respective service. If you do not specify anything under **Computer**, the setting **Prohibit all** will automatically become active when the configuration is saved.
Log	If **Log** is active, the control outputs a **red** message when a network packet for this service has been blocked. The control shows a blue message when a network packet for this service has been accepted.
Calculator	If you have selected the **Permit some** setting under **Method**, specify the computers here.
	The computers can be entered with their IP addresses or with a host name. Separate multiple computers by commas. If you use a host name, the system checks upon closing or saving the dialog whether this host name can be translated into an IP address. If this is not the case, an error message is displayed and the dialog will not terminate.
	If you specify a valid host name, this host name is translated into an IP address each time the control is started. If a computer that was entered with its name changes its IP address, you may have to restart the control or formally change the firewall configuration. In this case, the control uses the new IP address for a host name in the firewall.
Advanced options	These settings are only intended for your network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

9.3 Setting up data interfaces

Serial interfaces on the TNC 128

The TNC 128 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).

> **ℹ** 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 authentication.
>
> If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

Application

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.

To set up a data interface:

| MOD | ▶ Press the **MOD** key
 ▶ Enter the code number 123
 ▶ Define an interface in the machine parameter **RS232** (no. 106700)

Setting the RS-232 interface

To set up the RS 232 interface:
- ▶ Open the **RS232** folder
- > The control displays setting options with the following machine parameters.

Set BAUD RATE
(baud rate no. 106701)

Use the BAUD RATE to set the data transfer rate.

Enter a value between 110 and 115,200 baud.

**Set protocol
(protocol no. 106702)**

The communications protocol controls the data flow of a serial transmission.

> **ⓘ** Please note:
> - The **BLOCKWISE** setting designates a type of data transfer in which the data is transferred grouped in blocks.
> - The **BLOCKWISE** setting does **not** correspond to the data reception in blocks and the simultaneous execution of older controls in blocks. This function is no longer available for current controls.

Setting	Data transmission protocol
STANDARD	Standard data transmission Line-by-line transmission
BLOCKWISE	Packet-based data transfer
RAW_DATA	Transfer without protocol Transmission of characters only

**Set data bits
(dataBits no. 106703)**

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

**Check parity
(parity no. 106704)**

The parity bit defines whether transmission errors are checked.

The parity bit can take three different forms:

Setting	Meaning
NONE	No parity There is no error detection.
EVEN	Even parity There is an error if the receiver detects an uneven number of bits in his evaluation.
ODD	Odd parity There is an error if the receiver detects an even number of bits in his evaluation.

**Set stop bits
(stopBits no. 106705)**

The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission.

Set handshake
(flowControl no. 106706)

By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.

Setting	Meaning
NONE	No data flow control
	Handshake is not active
RTS_CTS	Hardware handshaking
	Transmission stopped by RTS active
XON_XOFF	Software handshaking
	Transfer stopped by DC3 (XOFF) active

File system for file operation
(fileSystem no. 106707)

In **fileSystem** you define the file system for the serial interface.

If you require no special file system, this machine parameter is not needed.

Setting	Meaning
EXT	Minimum file system for printers or non-HEIDEN-HAIN transmission software. It corresponds to the EXT1 and EXT2 operating modes on older HEIDENHAIN controls.
FE1	Communication with the TNCserver PC software or an external floppy disk unit

Block check character
(bccAvoidCtrlChar no. 106708)

The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection.

Setting	Meaning
TRUE	Ensures that the BCC does not correspond to a control character.
FALSE	Function not active

Condition of RTS line
(rtsLow no. 106709)

This optional parameter determines the level of the RTS line in the idle state.

Setting	Meaning
TRUE	The level is **low** in the idle state
FALSE	The level is **high** in the idle state

**Define behavior after receipt of ETX
(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).

Setting	Meaning
TRUE	The EOT character is not transmitted.
FALSE	The control transmits the EOT character.

Settings for transmission of data using TNCserver

When transmitting data with the TNCserver PC software, the machine parameter **RS232** (no. 106700) must be set to the settings below:

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

HEIDENHAIN data transfer software

Application

HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Prerequisites

System requirements for TNCremo:
- PC operating system:
  - Windows 8
  - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- A network connection to the control

Functionality

The TNCremo data transfer software provides the following areas:
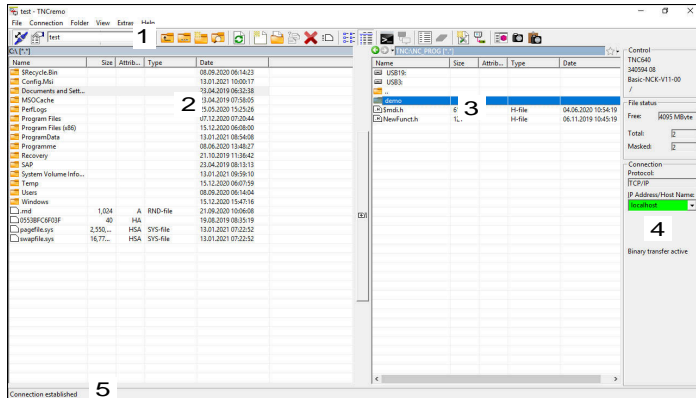
1 Toolbar

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.

Installing TNCremo

To install TNCremo on a PC:

▶ Start the SETUP.EXE installation program from the Explorer

▶ Follow the installer instructions

Starting TNCremo

To start TNCremo in Microsoft Windows 10:

▶ Press the Windows key

▶ Select the HEIDENHAIN folder

▶ Select TNCremo

  or

▶ Double-click the TNCremo desktop icon

Configuring a connection

Before connecting to the control, you need to configure a connection.

To configure a connection:

▶ Press the **Configure** button

> TNCremo opens the **Connection configuration** window.

▶ Select the **Management** tab

▶ Select **New...**

> TNCremo opens the **New configuration** window.

▶ Enter the connection name

▶ Press **OK**

> TNCremo automatically opens the **Connection** tab.

▶ Select the **Type of connection**

> **ℹ** If you use the default type of connection, a network connection (TCP/IP) is set up over the Ethernet interface.

▶ Select the **Settings** tab

▶ Enter the **IP address/Host name** of the control

▶ Select **OK**

> TNCremo saves the configuration.
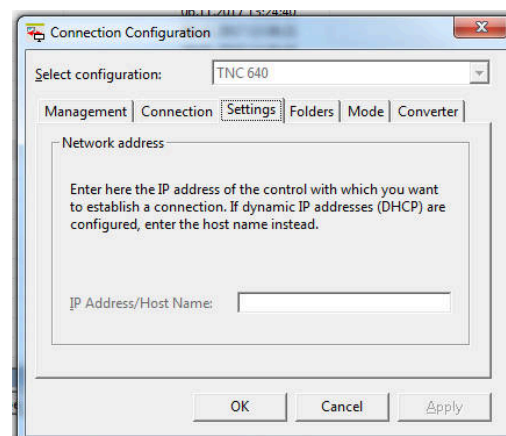
Establishing a connection to the control

Once you have configured a connection, you can connect your PC to the control.

To set up a connection to the control:

▶ Select the configured connection in the selection menu

▶ Select the **Set up connection** icon

> The connection to the control will be established.
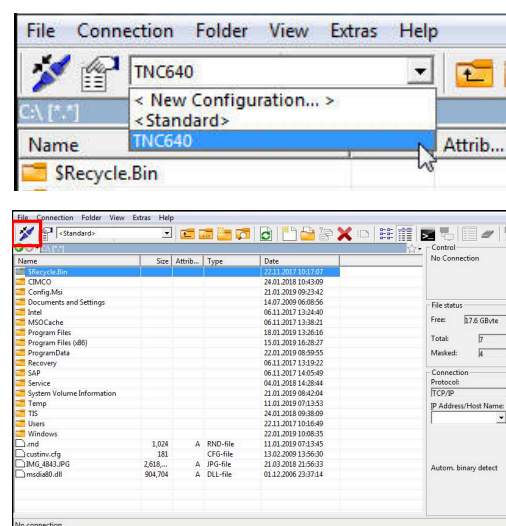
Switching the drive

You can switch the PC drive or the control drive that is shown in TNCremo.

To switch the shown drive:

▶ Select the PC's file list or the control's file list

▶ Select the **Change folder/drive** icon
> TNCremo opens a pop-up window.
▶ Select the desired drive in the selection menu
▶ Select **OK**
> TNCremo displays the selected drive.

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 authentication.

If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

■ You can download the current version of the TNCremo software from the **HEIDENHAIN website** for free.

9.4   Ethernet interface

Introduction

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network as a client.

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.

> ⓘ
> - 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.

Connection possibilities

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.

10BaseT / 100BaseTx / 1000BaseTx
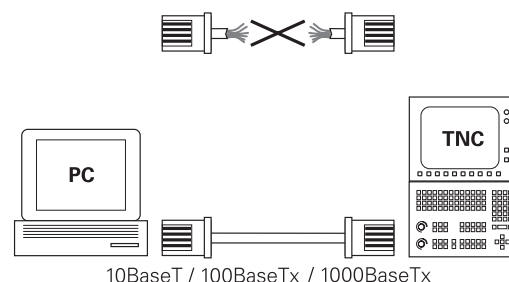
Ethernet connection icon

Icon	Meaning
	Ethernet connection
	The control displays the icon at the bottom right in the taskbar.
	Further information: "Overview of taskbar", Page 289
	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

The Network settings window

In the **Network settings** window you define the settings for the control's Ethernet interface.

> ℹ️ Have a network specialist configure the control.
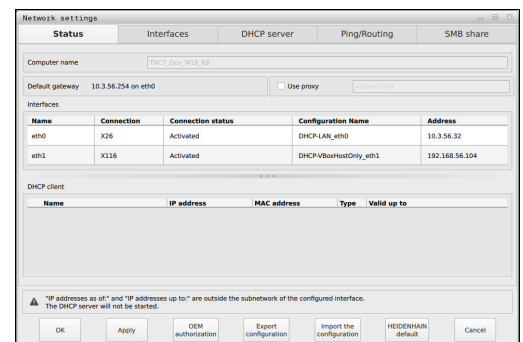
Status tab

The **Status** tab contains the following information and settings:

Domain	Information or Setting
Domain	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.

Domain	Information or Setting
Interfaces	The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty. The control displays the following information in the table: ■ **Name** (e.g., **eth0**) ■ **Connection** (e.g., **X26**) ■ **Connection status** (e.g., **CONNECTED**) ■ **Configuration Name** (e.g., **DHCP**) ■ **Address** (e.g., **10.7.113.10**) **Further information:** "The Interfaces tab", Page 315
DHCP client	The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty. The control displays the following information in the table: ■ **Name** Host name and connection status of the device. The control shows the following connection status: ■ Green: Connected ■ Red: No connection ■ **IP address** Dynamically assigned IP address of the device ■ **MAC address** Physical address of the device ■ **Type** Type of connection The control displays the following connection types: ■ **TFTP** ■ **DHCP** ■ **Valid up to** Time until which the IP address is valid without being renewed The machine manufacturer can make settings for these devices. Refer to your machine manual.

The Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.

The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a toggle switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ **CONNECTED** Connected ■ **DISCONNECTED** Connection separated ■ **CONFIGURING** The IP address is being fetched from the server ■ **NOCARRIER** No cable present
Configuration Name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ **DHCP-LAN**: Settings for the standard interface for a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 318

ⓘ
- If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.
- The control exclusively supports the **Ethernet** connection type.

The DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).

Refer to your machine manual.

The Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

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.

The SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station.

When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.

> **ℹ** **Further information:** Programming station for milling controls
>
> You download the documentation together with the programming station software.

Opening the Network settings window

To open the general network settings:

| MOD |

▶ Press the **MOD** key

▶ Enter the code number NET123.

| PGM MGT |

▶ Press the **PGM MGT** key

| NET |

▶ Press the **NET** soft key

| CONFIGURE NETWORK |

▶ Press the **CONFIGURE NETWORK** soft key

> The control opens the **Network settings** window.

Exporting and importing a network profile

To export a network profile:

▶ Opening the **Network settings** window

▶ Select **Export configuration**

> The control opens a window.

▶ Select the storage location for the network profile (e.g., **TNC:/etc/sysconfig/net**)

▶ Select **Open**

▶ Select the desired network profile

▶ Select **Export**

> The control saves the network profile.

ℹ You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:

▶ Open the **Network settings** window

▶ Select **Import the configuration**

> The control opens a window.

▶ Select the storage location of the network profile

▶ Select **Open**

▶ Select the desired network profile

▶ Press **OK**

> The control opens a window with a prompt.

▶ Press **OK**

> The control imports and activates the selected network profile.

▶ You might need to restart the control

ℹ Use the **HEIDENHAIN presettings** button to import the default values of the network settings.

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: "Changing the HEROS conversational language", Page 370

Network configuration with Advanced Network Configuration

Application

Use the **Advanced Network Configuration** to add, edit or remove profiles for the network connection.

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.
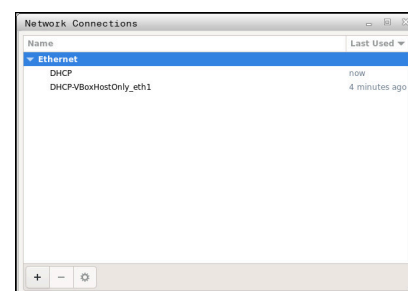
The **Network Connections** window

Symbols in the **Network connections** window
The following symbols are shown in the **Network Connections** window:

Icon	Function
✚	Add network connection
➖	Remove network connection
⚙	Edit network connection The control opens the **Editing network connection** window. **Further information:** "The Editing network connection window", Page 319

When a connection is active, the control displays a symbol in the header:

Icon	Meaning
⇄🔒	**Secure connection configuration** External access to the control is active; all connections are using a secure connection configuration.
⇄🔓	**Non-secure connection configuration** An external access to the control is active but at least one connection is using a non-secure connection configuration.

Further information: "Display screen", Page 63

The Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.
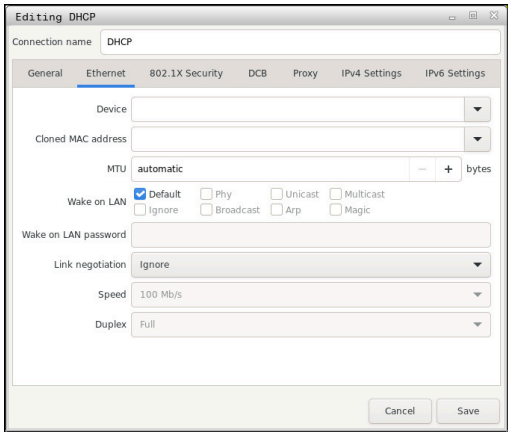
The **Editing network connection** window

The **General** tab
The **General** tab contains the following settings:

Setting	Meaning
Connect automatically	If you are using several profiles, you can define an order of priority for the connection here.
	The control connects the network with the highest priority first.
	Input: **-999…999**
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN when using this connection	Currently no function
Bonded connections:	Currently no function

The **Ethernet** tab
The **Ethernet** tab contains the following settings:

Setting	Meaning
Service:	Here you can select the Ethernet interface.
	If you do not select an Ethernet interface, this profile can be used for any Ethernet interface.
	Selection by means of a selection window
Cloned MAC address:	Currently no function
_MTU:	Here you can define the maximum package size in bytes.
	Input: **Automatic**, **1...10000**
_Private key password:	Currently no function
Wake-on-LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection:
	■ **Ignore**
	Retain the configurations already existing on the device.
	■ **Automatic**
	The speed and duplex settings are configured automatically for the connection.
	■ **Manual**
	Configure the speed and duplex settings for the connection manually.
	Selection by means of a selection window
Speed	Here you have to select the speed settings:
	■ **10 Mb/s**
	■ **100 Mb/s**
	■ **1 Gb/s**
	■ **10 Gb/s**
	Only if **Link negotiation** has been selected **Manual**
	Selection by means of a selection window
Full duplex	Here you have to select the duplex setting:
	■ **Half**
	■ **Full**
	Only if **Link negotiation** has been selected **Manual**

Setting	Meaning
	Selection by means of a selection window

The **802.1X Security** tab
Currently no function

The **DCB** tab
Currently no function

The Proxy tab
Currently no function

The **IPv4 Settings** tab
The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
_Method:	Here you have to select a network connection method: ■ **Automatic (DHCP)** If the network uses a DHCP server for IP address assignment ■ **Automatic (DHCP) addresses only** If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually ■ **Manual** Assign the IP address manually ■ **Link-Local Only** Currently no function ■ **Shared to other computers** Currently no function ■ **Disabled** Deactivate IPv4 for this connection
Automatic, addresses only	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with **_Method: Manual**
Additional DNS servers:	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with **_Method: Manual** and **Automatic (DHCP) addresses only**
Additional search domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **_Method: Manual**
DHCP client ID:	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

The **IPv6 Settings** tab
Currently no function

Settings for network drives

ℹ️ Have a network specialist configure the control.

You can connect network drives to the control. If the control is connected to a network and the file shares are connected, the control displays additional drives in the directory window of the file management

In the **Network drive** area of the **Mount Setup** window, the control shows a list of all defined network drives and the status of each drive.

You can define any number of network drives, but only seven can be connected at a time

In the **Status Log** area, the control shows status information and error messages.

Open settings

To open the settings for network drives:

| PGM MGT | ▶ Press the **PGM MGT** key |

| NET | ▶ Press the **NET** soft key |

| DEFINE NETWORK CONNECTN. | ▶ Press the **DEFINE NETWORK CONNECTN.** soft key |
| | > The control opens the **Mount Setup** window. |
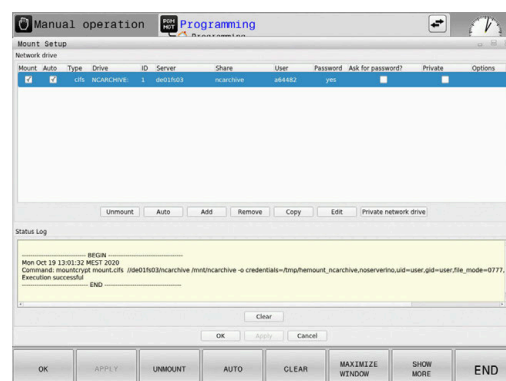
Overview of the soft keys

Soft key	Button	Meaning
MOUNT	**Mount**	Connect a network drive The control ticks the check box in the **Mount** column when there is an active connection.
UNMOUNT DEVICE	**Unmount**	Disconnect a network drive
AUTO	**Auto**	Automatically connect the network drive whenever the control is switched on The control ticks the check box in the **Auto** column when there is an automatic connection.
ADD	**Add**	Define a new network drive
REMOVE	**Remove**	Delete an existing network drive
COPY	**Copy**	Copy a network drive
EDIT	**Edit**	Editing a network drive
CLEAR	**Clear**	Delete the contents of the **Status Log** area
PRIVATE	**Private network drive**	User-specific network drive when user administration is active The control ticks the check box in the **Privat** column when there is a user-specific connection.

Adding a network drive

Requirements

Prerequisites for connecting a network drive:

- Connection with the network
- The control must reach the server in the network
- The credentials and the drive path are known

Adding a network drive

To add a network drive:

▶ Select **Add**
> The control opens the **Mount assistant** window.
▶ Define the settings on the individual tabs
▶ Select **Forward** after each tab
▶ On the **Check** tab, check the settings
▶ Select **Apply**
> The control connects the network drive.
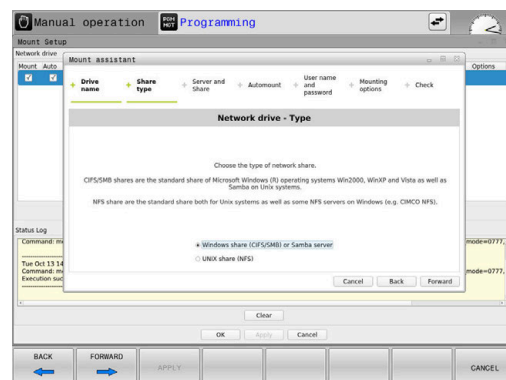
Settings for the network drive

The control guides you through the settings with the **Mount assistant**.

Tab	Setting
Drive name	■ **Drive name:** Network drive name shown in the file management of the control The names must be all uppercase letters, terminated by a colon (**:**). ■ **Private network drive** With user administration active, the connection is only visible to the user who created it. ℹ️ To create and edit public connections, you need the HEROS.SetShares right. Without this right, users are able to start and terminate public connections but can create and edit only private connections. **Further information:** "Definition of roles", Page 351
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address of network drive ■ **Share name:** Designation used to share the folder accessed by the control

Tab	Setting
Automount	**Connect automatically (not possible with the "Ask for password?" option)** The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ **Single Sign On** With user administration active, the control automatically connects an encrypted network drive when the user logs in. ■ **Windows user name:** ■ **Ask for password? (not possible with the "Connect automatically" option)** Select whether a password is required upon connecting. ■ **Password** ■ **Password verification**
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection
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 NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input range 512 to 8192
wsize=4096	Packet size in bytes for data transmission Input range 512 to 8192
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will repeat the attempt
nfsvers=2	Protocol version ⓘ If you use the CIMCO NFS software, you must set this option. CIMCO NFS supports NFS only up to version 2.

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to include the domain in the user name, but rather specify it as an option.
vers=3.1.1	Protocol version
sec=ntlmssp	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.

ⓘ To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

Depending on the software status of the control, the network drive may require an older protocol version. In this case, you may change the protocol version with the **vers=** auxiliary parameter. Contact your network specialist.

9.5 File transfer with SFTP (SSH File Transfer Protocol)

Application

SFTP (SSH File Transfer Protocol) provides a secure way to connect client applications to the control and to transfer files at high speed from a PC to the control. The connection is routed via an SSH tunnel.

Related topics

■ User administration

Further information: "User administration", Page 333

■ Principle of the SSH connection

Further information: "User authentication from external applications", Page 356

■ Firewall settings

Further information: "Firewall", Page 302

Requirements

■ PC software TNCremo with version 3.3 or higher is installed

Further information: "HEIDENHAIN data transfer software", Page 308

■ **SSH** service is permitted in the firewall of the control

Further information: "Firewall", Page 302

Description of function

SFTP is a secure transmission protocol supported by various operating systems for client applications.

To set up the connection, you need a key pair consisting of a public and a private key. You transfer the public key to the control and assign it to a user through the user administration. The private key is required by the client application to set up a connection to the control.

HEIDENHAIN recommends using the CreateConnections application to generate the key pair. CreateConnections is installed together with the PC software TNCremo with version 3.3 and higher. CreateConnections lets you transfer the public key directly to the control and assign it to a user.

You can also use other software to generate the key pair.

Setting up an SFTP connection with CreateConnections

For an SFTP connection using CreateConnections, the following are required:

- Connection with secure protocol, such as **TCP/IP Secure**
- User name and password of the desired user are known

> **ℹ** When you transfer the public key to the control, you must enter the user's password twice.
>
> If user administration is inactive, the user **user** is logged in. The password for the user **user** is **user**.

To set up an SFTP connection with CreateConnections:

- ▶ Press the **DIADUR** key to open the **HEROS menu** menu
- ▶ Select **Settings**
- ▶ Select **Current User**
- > The control opens the **Active user** window.
- ▶ Select **Certificate and keys**
- > The control opens a pop-up window.
- ▶ Select the **Allow password authentication** check box
- ▶ Select **Store and restart server now**
- ▶ Create a key pair with CreateConnections and transfer it to the control

> **📖** For more information, refer to the integrated help system of TNCremo.
>
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

- ▶ Deactivate the check box **Allow password authentication**
- ▶ Select **Store and restart server now**
- ▶ Select **END**
- ▶ Select **Close**
- > The control closes the **Active user** window.
- ▶ Transfer the private key to the client application
- ▶ Connect the client application to the control

> **📖** Please note the documentation of the client application.

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 authentication. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

- During the connection, the rights of the user to whom the used key is assigned are active. The directories and files displayed, as well as the access options, vary depending on the permissions.

- You can also transfer a public key to the control by using a USB device or network drive. In this case, you do not need to activate the **Allow password authentication** check box.

- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

9.6   SELinux security software

SELinux is an extension for Linux-based operating systems. **SELinux** is an additional security software in the sense of Mandatory Access Control (MAC) and protects the system against the execution of unauthorized processes or functions, and therefore against viruses and other malware.

MAC means that every action must be explicitly permitted, otherwise it will not be executed by the control. The software is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can only be executed if the standard functions and access control of **SELinux** permit it.

ⓘ   The SELinux installation of the control has been prepared to permit running only programs installed with the HEIDENHAIN NC software. Other programs cannot be run with the standard installation.

The access control of **SELinux** under HEROS 5 is regulated as follows:

- The control executes only applications that are installed with the HEIDENHAIN NC software
- Files in connection with the security of the software (**SELinux** system files, HEROS 5 boot files, etc.), may only be modified by programs that have been selected explicitly.
- New files generated by other programs must never be executed
- USB data carriers cannot be deselected
- There are only two processes that are permitted to execute new files:
    - Starting a software update: A software update from HEIDENHAIN can replace or change system files
    - Starting the SELinux configuration: The configuration of **SELinux** is usually password-protected by your machine manufacturer; refer to the relevant machine manual.

ⓘ   HEIDENHAIN recommends activating **SELinux** because it provides additional protection against attacks from outside.

9.7 User administration

Introduction

⚙️ Refer to your machine manual.

Some user administration areas are configured by the machine manufacturer.

If you want to use user administration on a control without HEIDENHAIN keyboard, you need to connect an external alphabetic keyboard to the control.

User administration is inactive in the control's factory default setting. This status is called **legacy mode**. In **legacy mode**, the control's behavior is identical to that of older software versions without user administration.

It is not mandatory that you use user administration, but it is essential for the implementation of an IT security system.

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
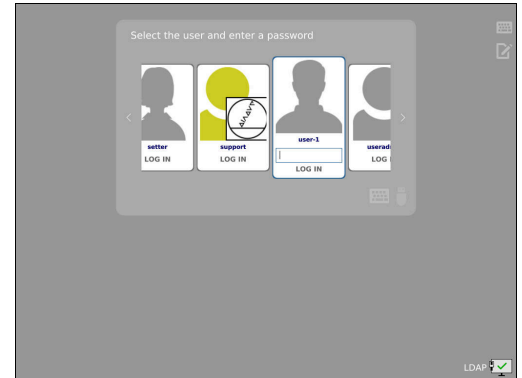
User administration enables you to define users with different access rights.

The following options are available for saving your user data:

- **Local LDAP database**
    - 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
      Further information: "Local LDAP database", Page 338
- **LDAP on remote computer**
    - Importing an LDAP server configuration file
      Further information: "LDAP on remote computer", Page 339
- **Connection to Windows domain**
    - Integrating user administration on more than one control
    - Using different roles on different controls
      Further information: "Logon to Windows domain", Page 340

ℹ️ Parallel operation of Windows users and users from an LDAP database is possible.

Configuring user administration

User administration is inactive in the control's factory default setting.
This status is called **Legacy-Mode**.

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

1 Calling user administration
2 Activating user administration
3 Creating a **useradmin** user
4 Setting up a database
5 Creating further users
  Further information: "Creating further users", Page 344

Calling user administration

To call user administration:

▶ Press the **DIADUR** key to open the **HEROS menu** menu
▶ Select the **Settings** menu item
▶ Select the **UserAdmin** menu item
> The control opens the **User administration** window.

> **i**   You can exit the **User administration** window after
>   each configuration step.
>
>   If you exit the **User administration** window directly
>   after having activated user administration, the control
>   will prompt you for a restart once.

Activating user administration

To activate user administration:

▶ Call user administration

▶ Press the **User administration active** soft key

＞ The control shows the message **Password for user 'useradmin' missing**.
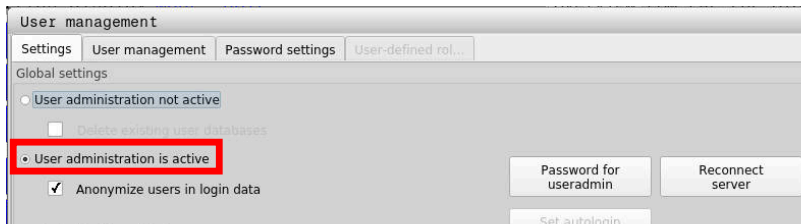
> ℹ The purpose of the **Anonymize users in log data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in log data** function, the system will show personalized user data in all control log files. If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case, it is your responsibility to ensure that all required data protection provisions have been made at your company.

▶ Retain or reactivate the active status of the **Anonymize users in log data** function

---

Deactivating user administration

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.

User administration can only be deactivated by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "HEIDENHAIN function users", Page 350

To deactivate user administration:

▶ Log in as the appropriate function user
▶ Call user administration
▶ Select **User administration inactive**
▶ If necessary, check **Delete existing user databases** to delete any configured users and user-specific directories

APPLY	▶ Press the **APPLY** soft key
END	▶ Press the **END** soft key

     > The control opens the **System reboot required** window.
     ▶ Select **Yes**
     > The control triggers a restart.

Creating a useradmin function user

After user administration has been activated once, create the **useradmin** function user.

The **useradmin** user is comparable to the local administrator of a Windows system.

To create the **useradmin** user:

▶ Select **Password for useradmin**
> The control opens the **Password for user 'useradmin'** pop-up window.
▶ Set the password for the **useradmin** user
▶ Select **Set new password**
> The control shows the message **Settings and password for 'useradmin' were changed**.

i   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.

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

i   The **useradmin** user automatically receives the HEROS.Admin role. This allows him to manage users in user administration, provided that he knows the LDAP database password. The **useradmin** user is a function user who has been pre-defined by HEIDENHAIN. You can neither add roles to function users, nor delete their roles.

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.

Setting up a database

To set up the database:

▶ Select the database for saving the user data

▶ Set up a database

▶ Press the **APPLY** soft key

▶ Press the **END** soft key

> The control opens the **System reboot required** window.

▶ Press **Yes** to restart the system

> The control restarts.
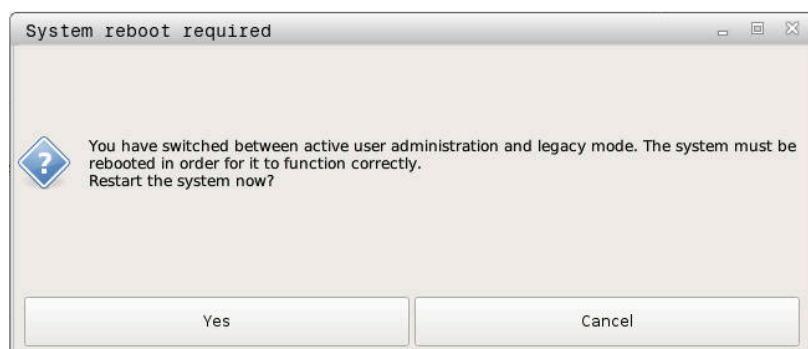
Local LDAP database

Before you can use the **Local LDAP database** function, the following requirements must be met:

▪ The user administration is active

▪ The **useradmin** user has been configured

To set up a **Local LDAP database**:

▶ Call the user administration

▶ Select the **LDAP user database** function

> The control enables the dimmed area for editing the LDAP user database.

▶ Select the **Local LDAP database** function

▶ Select the **Configuration** function

> The control opens the **Configure local LDAP database** window.

▶ Enter the name of the **LDAP domain**

▶ Enter the password

▶ Repeat the password

▶ Press the **OK** soft key

> The control closes the **Configure local LDAP database** window.
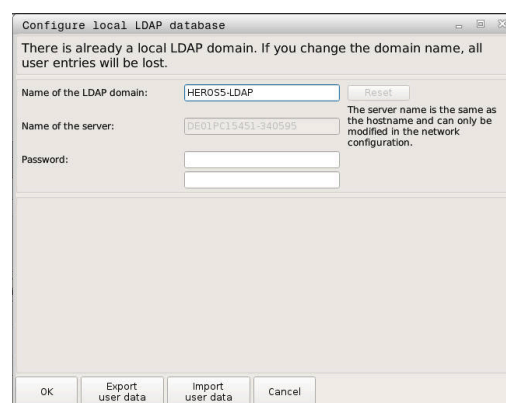
🛈   Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.

        Passwords must not be trivial and must be known only to the administrators.

        Further information: "Creating further users", Page 344

🛈   If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

LDAP on remote computer

Requirements

Before you can use the **LDAP on remote computer** function, the following requirements must be met:

- User administration is active
- The **useradmin** user has been configured
- An LDAP database has been set up in the company network
- A server configuration file of an existing LDAP database must be stored on the control or on a PC in the network
- The PC with the existing configuration file is running.
- The PC with the existing configuration file is accessible in the network

Providing a server configuration file

Provide a server configuration file for an LDAP database as follows:

- ▶ Call the user administration
- ▶ Select the **LDAP user database** function
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select the **Local LDAP database** function
- ▶ Select the **Export server configuration** function
- > 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 server configuration file was exported successfully.

Using the LDAP database on another computer

To use the **LDAP on remote computer** function:

- ▶ Call the user administration
- ▶ Select the **LDAP user database** function
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select the **LDAP on remote computer** function
- ▶ Select the **Import server configuration** function
- > The control opens the **Import LDAP configuration file window.**
- ▶ Select the existing configuration file
- ▶ Select **FILE**
- ▶ Press the **APPLY** soft key
- > The configuration file was imported.
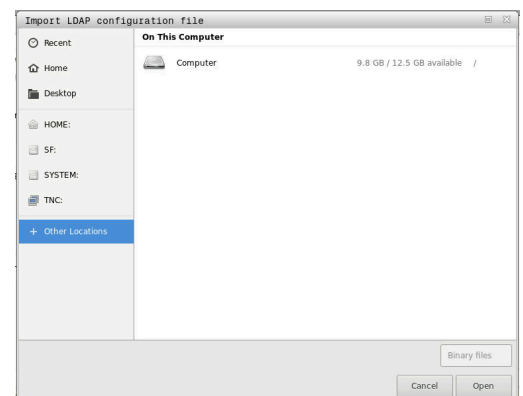
Logon to Windows domain

Prerequisites

Before you can use the **Connection to Windows domain** function, the following requirements must be met:

- User administration is active
- The **useradmin** user has been configured
- There is a Windows Active Domain Controller in the network
- The domain controller is accessible in the network
- Organizational unit for HEROS roles known
- For logon with computer account:
  - You have access to the password of the domain controller
  - You have access to the user interface of the domain controller or you are supported by an IT administrator
- For logon with function user:
  - User name of the function user
  - Password of the function user

Joining a Windows domain with a computer account

To join a Windows domain with a computer account:

▶ Call the user administration
▶ Select the **Connection to Windows domain** function
▶ Select the **Join Active Directory domain (with computer account)** check box
▶ Select the **Find domain** function

> **i** Use the **Configuration** function to define various settings of your 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

▶ Press the **APPLY** soft key
> The control opens the **Connect to domain** window.

> **i** The **Organizational unit for computer account:** function allows you to specify in which of the already existing organizational units you want to create the access, such as
>
> ■ 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
> 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.

> ⓘ 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:
>   ▶ Press the **Add role definition** soft key
>   ■ Select the **Add** function
>   You can enter the roles directly into the domain in
>   this case.
>   ■ Press **Export**
>   With this function, you can export the roles to a
>   file in .ldif format.

> All of the required roles have been created in the domain as
  groups.

Joining a Windows domain with a function user

To join a Windows domain with a function user:
▶ Opening the **User administration** window
▶ Select **Connection to Windows domain**
▶ Select the **Active Directory with function user** check box
▶ Select **Find domain**
> The control selects a domain.
▶ Select **Configuration**
▶ Check the data for **Domain name:** and **Key Distribution Center
  (KDC):**
▶ Enter **Organizational unit for HEROS roles:**
▶ Enter the user name and password of the function user
▶ Press **OK**
▶ Select **APPLY**
> The control connects to the Windows domain found.
> The control checks whether all of the required roles have been
  created in the domain as groups.

Creating groups

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:
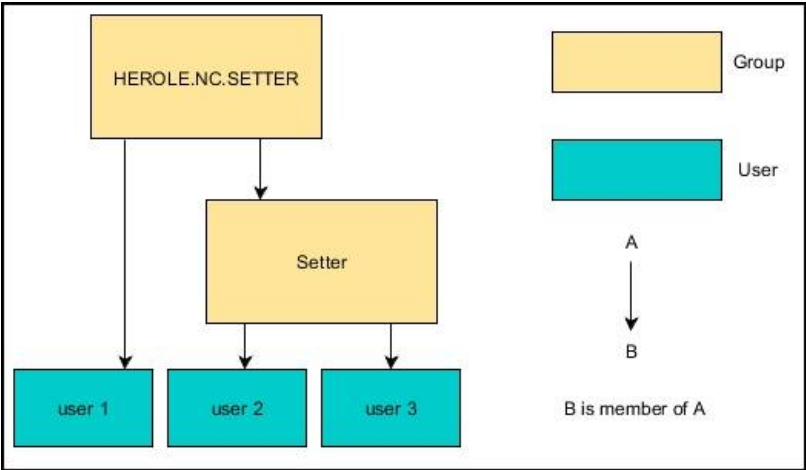
Example 2

Users from various sectors (workshops) are members of groups with different prefixes:
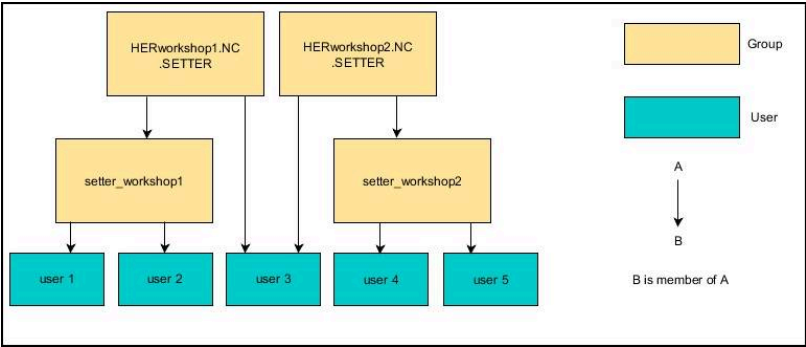
Exporting and importing a Windows configuration file

If you have connected the control to the Windows domain, you can export the required configurations for other controls.

To export the Windows configuration file:

▶ Open the **User administration** window
▶ Select **Connect to Windows domain**
▶ Select **Export the Windows config.**
> The control opens the **Export the Windows domain configuration** window.
▶ Select the directory for the file
▶ Enter the name for the file
▶ Select the **Export the function user's password?** check box, if required
▶ Select **Export**
> The control saves the Windows configuration as a BIN file.

To import the Windows configuration file of another control:

▶ Open the **User administration** window
▶ Select **Connect to Windows domain**
▶ Select **Import the Windows config.**
> The control opens the **Import the Windows domain configuration** window.
▶ Select the existing configuration file
▶ Select the **Import the function user's password?** check box, if required
▶ Select **Import**
> The control adopts the configurations for the Windows domain.

Creating further users

Before you can create further users, the following requirements must be met:

■ User administration has been configured
■ LDAP database has been selected and configured

> 🛈 The **User administration** tab is only active for the following databases:
> ■ **Local LDAP database**
> ■ **LDAP on remote computer**
> If **Connection to Windows domain** is used, you need to configure the users in the Windows domain.
> **Further information:** "Logon to Windows domain", Page 340

Open the User administration tab

To manage users:

▶ Call user administration

▶ Select the **User administration** tab

▶ Press the **EDIT ON** soft key

> The control prompts you to enter the password of the user database, if applicable.

> After you have entered the password, the control opens the **User administration** menu.

You can edit the existing users or create new users.
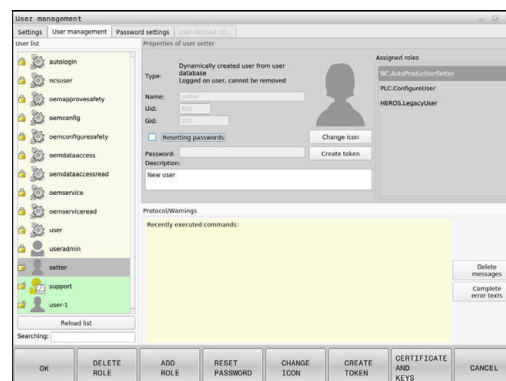
Creating a new user

To create a new user:

▶ Press the **Create new user** soft key

> The control opens a window for creating a user.

▶ Enter the user name

▶ Enter the password for the user

> **ℹ** The user must change the password when logging in for the first time.
> **Further information:** "Login to user administration", Page 360

▶ Create a user description, if desired

▶ Press the **Add role** soft key

▶ In the selection window, select the roles corresponding to the user
Further information: "Definition of roles", Page 351

▶ Press the **Add** soft key

> **ℹ** Two additional soft keys are available in the menu:
> ■ **Add external login**
>   Adds Remote.HEROS.Admin in place of HEROS.Admin, for example.
>   This role is only enabled for remote logon to the system.
> ■ **Add local login**
>   Adds Local.HEROS.Admin in place of HEROS.Admin, for example.
>   This role is only enabled for local logon at the control's screen.

▶ Press the **CLOSE** soft key

> The control closes the window for creating a user.

▶ Press the **OK** soft key

▶ Press the **APPLY** soft key

> The control adopts the changes.

▶ Press the **END** soft key

> The control closes user administration.

> **ℹ** If you have not restarted your control after configuring the database, the control prompts you to restart for the changes to take effect.
> **Further information:** "Configuring user administration", Page 334
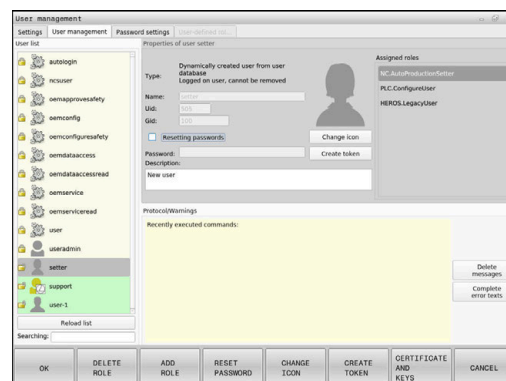
Adding profile images

You can optionally also assign images to the users. The **Default user icons:** from HEIDENHAIN are available for this purpose. You can also load your own images in JPEG or PNG format to the control. Then you can use these image files as profile images.

To define profile images:

▶ Log on a user with the HEROS.Admin role, e.g. **useradmin**

   Further information: "Login to user administration", Page 360

▶ Call user administration

▶ Select the **User administration** tab

▶ Press the **Edit user** soft key

▶ Press the **Change icon** soft key

▶ Select the desired image in the menu

▶ Press the **Choose icon** soft key

▶ Press the **OK** soft key

▶ Press the **APPLY** soft key

> The control adopts the changes.

> 🛈 You can also directly add profile images when creating the users.

Password settings in user administration

The Password settings tab

Users with the HEROS.Admin role can specify exact requirements for user passwords on the **Password settings** tab.

Further information: "Rights", Page 354

> 🛈 If you do not comply with the requirements defined here when creating your password, the control will issue an error message.

To open the **Password settings** tab:

▶ Log on as a user with HEROS.Admin role

▶ Call user administration

▶ Select the **Password settings** tab

▶ Press the **EDIT ON** soft key

> The control opens the **Enter password of LDAP database** window.

▶ Enter password

> The control releases the **Password settings** tab for editing.
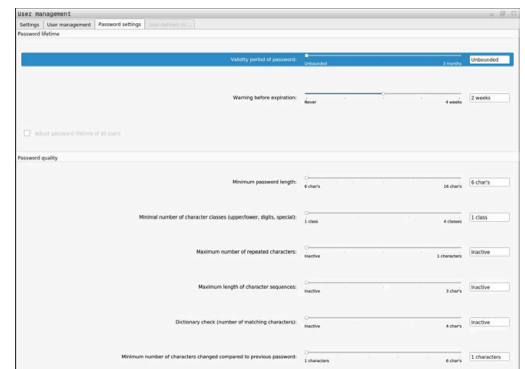
Defining password settings

The control allows you to configure user password requirements by setting various parameters.

To edit the parameters:

▶ Open the **Password settings** tab
▶ Select the desired parameter
> The control highlights the selected parameter in blue.
▶ Set the desired parameter using the slider
> The control shows the selected parameter in the display field.

APPLY	▶ Press the **APPLY** soft key
	> The control applies the change.

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.

Access rights

The user administration is based on the Unix rights management.
Access to the control is controlled by means of rights.

The user administration differentiates between the following terms:

- User
- Roles
- Rights
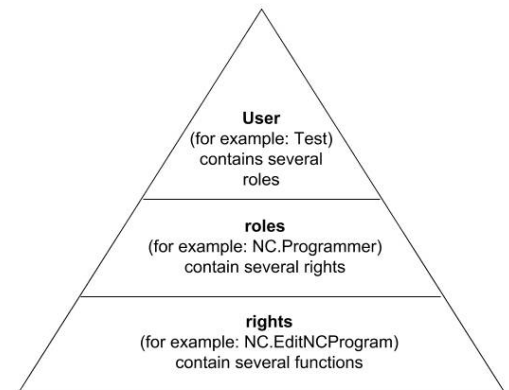
User

A user can be pre-defined in the control or he can be defined by the
end user.

The user administration offers the following types of users:

- Function users pre-defined by HEIDENHAIN
  Further information: "HEIDENHAIN function users", Page 350
- Function users pre-defined by the machine manufacturer
- Self-defined users

A user contains all the roles that have been assigned to him.

> ⓘ Your machine manufacturer defines function users who are
> required for specific tasks such as machine maintenance.
>
> Depending on the task assigned, you can use one of the
> pre-defined function users or you have to create a new user.
>
> The access rights of HEIDENHAIN function users are
> already pre-defined in the control's factory default setting.

Roles

Roles consist of a combination of rights that cover certain functions
provided by the control.

- **Operating system roles:**
- **NC operator roles:**
- **Machine tool builder (PLC) roles:**

All of these roles are pre-defined in the control.

You can assign more than one role to a user.

Rights

Rights consist of a combination of functions that cover a task area of the control, e.g. editing the tool table.

- 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.

> **i** Ensure that every user is assigned all access rights he needs. The access rights result from the actions a user performs on the control.

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.

- **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.

- **Function users defined by the machine manufacturer**

  > **⚙** Refer to your machine manual.
  >
  > The users defined by the machine manufacturer can differ from those pre-defined by HEIDENHAIN.
  >
  > The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.
  >
  > By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.
  >
  > **Further information:** "Current User", Page 367

- **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**.

- **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.

Definition of 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.

> **i** Every user should have at least one role from the operating system area and at least one role from the programming area.
>
> 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.
>
> You can therefore also make the rights of a user dependent on the access used to operate the control.
>
> 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).

Advantages of classification in roles:
- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

> **i** Various applications require access to certain interfaces. Where required, the administrator also needs to set up rights to access the required interfaces as well as rights to access certain functions and additional programs. These rights are granted by the **Operating system roles:**.

> **i** The following contents can change in the following software versions of the control:
> - HEROS role names
> - Unix groups
> - Basic ID number

Operating system roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	▪ HEROS.MountShares	▪ mnt	▪ 335
	▪ HEROS.Printer	▪ lp	▪ 9
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	▪ HEROS.SetShares	▪ mntcfg	▪ 334
	▪ HEROS.ControlFunctions	▪ ctrlfct	▪ 340
HEROS.LegacyUser	With the **LegacyUser** role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. User administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	▪ HEROS.BackupUsers	▪ userbck	▪ 337
	▪ HEROS.PrinterAdmin	▪ lpadmin	▪ 16
	▪ HEROS.ReadLogs	▪ logread	▪ 342
	▪ HEROS.SWUpdate	▪ swupdate	▪ 341
	▪ HEROS.SetNetwork	▪ netadmin	▪ 336
	▪ HEROS.SetTimezone	▪ tz	▪ 333
	▪ HEROS.VMSharedFolders	▪ vboxsf	▪ 1000
HEROS.LegacyUserNoC-trlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically.		
	This role grants the rights of the LegacyUser role, with the exception of the following right:		
	▪ HEROS.ControlFunctions	▪ ctrlfct	▪ 340
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the **LegacyUser** role, as well as the following rights:		
	▪ HEROS.BackupMachine	▪ backup	▪ 338
	▪ HEROS.UserAdmin	▪ useradmin	▪ 339

NC operator roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
NC.Operator	This role allows you to run NC programs.		
	■ NC.OPModeProgramRun	■ 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
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	■ NCApproveFsAxis	■ 319
	■ NC.EditPocketTable	■ NCEdPocket	■ 307
	■ NC.SetupDrive	■ NCSetupDrv	■ 315
	■ NC.SetupProgramRun	■ NCSetupPgRun	■ 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	■ NCSchedulePgRun	■ 304
NC.LegacyUser	With the **LegacyUser** role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The **LegacyUser** has the same rights as the AutoProductionSetter.		
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	■ NCEditNCPgmAdv	■ 327
	■ NC.EditTableAdv	■ NCEditTableAdv	■ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	■ NC.RemoteProgramRun	■ NCRemotePgmRun	■ 329

Machine manufacturer (PLC) roles:

Role	Privileges		
	HEROS role name	**UNIX group**	**Basic ID number**
PLC.ConfigureUser	This roles grants the rights on code number **123**.		
	▪ NC.ConfigUserAdv ▪ NC.SetupDrive	▪ NCConfigUserAdv ▪ NCSetupDrv	▪ 316 ▪ 315
PLC.ServiceRead	This role allows read-only access during servicing. This role can be used to display various types of diagnostic information		
	▪ NC.Data.AccessServiceRead	▪ NCDAServiceRead	▪ 324

> ⚙ Refer to your machine manual.
> The machine manufacturer can adapt the PLC roles.
>
> When the **Machine manufacturer (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:
> ▪ The names of the roles
> ▪ The number of roles
> ▪ The functionality of the roles

Rights

The table below lists all of the individual rights.

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.
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.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 role name	Description
HEROS.SetTimezone	Adjustment of date and time, time zone and time synchronization via NTP and the **HEROS menu**.
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.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.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.OpcUaOEMConfigured-DataRead	Read-access through OPC UA NC Server to data defined by the machine manufacturer

Enabling Autologin

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.

For further authorizations, the control still requires an authentication to be entered.

In order to enable **Autologin**, the following requirements must be met:

■ User administration has been configured
■ The user for **Autologin** has been defined

To enable the **Autologin** function:

▶ Call user administration
▶ Select the **Settings** tab
▶ Press the **Global settings** soft key
▶ Select the **Enable autologin** check box
> The control opens a window for selecting the user.
▶ Select the user
▶ Enter the user's password
▶ Press the **OK** soft key

User authentication from external applications

Introduction

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.

> **i**   The the encryption used with the SSH tunnel protects the communication from attackers.

Concept of transmission through an SSH tunnel

Prerequisites:

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
  - Private key
  - Public key

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.
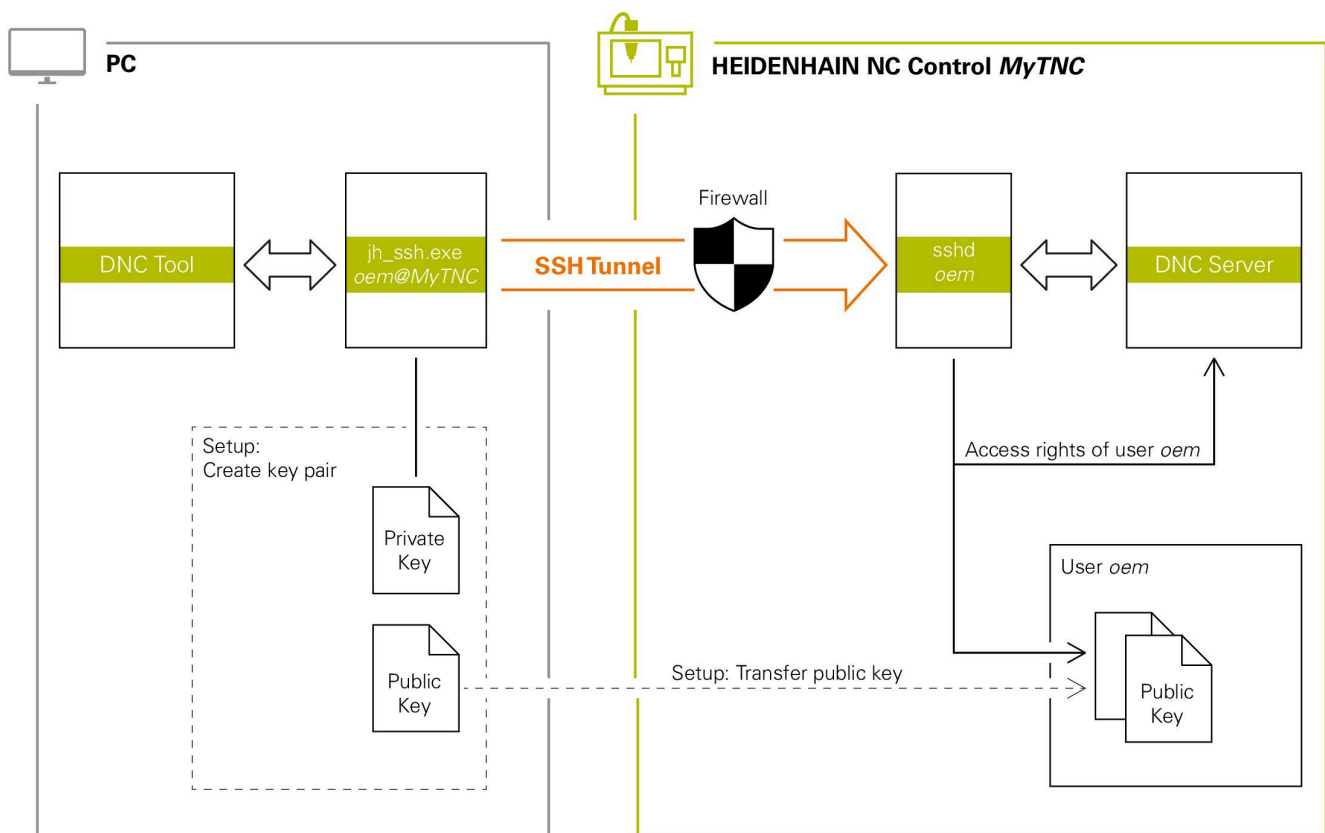
Use in external applications

> ℹ️ 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 authentication.
>
> If user administration is inactive, the control also
> automatically blocks non-secure LSV2 or RPC connections.
> In the optional machine parameters **allowUnsecureLsv2**
> (no. 135401) and **allowUnsecureRpc** (no. 135402), the
> machine manufacturer can define whether the control
> will permit non-secure connections. These machine
> parameters are included in the **CfgDncAllowUnsecur**
> (no. 135400) data object.
>
> **Further information:** "Serial interfaces on the TNC 128",
> Page 305

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.

> ℹ️ Once the connection configurations have been set up,
> they can be shared among all HEIDENHAIN PC tools for
> establishing a connection.

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.

> ℹ️ 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.

Setting up and removing a secure connection

To set up a secure connection for the logged-on user:

▶ Select the **Machine settings** group in the MOD menu
▶ Select the **External access** function
▶ Press the **Key management** soft key
> The control opens the **Certificate and keys** window.
▶ Select the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
▶ Use **TNCremo** to set up the secure connection (TCP secure).

> For details, refer to the integrated help system of TNCremo.

> The public key has been stored on the control by TNCremo.

> In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

▶ Deselect the **Allow password authentication** function
▶ Press the **Store and restart server now** soft key
> The control has adopted your changes.

> ■ In addition to using the PC tools for setup with password authentication, you can also import the public key into the control by using a USB stick 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 having to transfer them to the control.

To delete a key in the control so that a user can no longer use the secure connection:

▶ Select the **Machine settings** group in the MOD menu
▶ Select the **External access** function
▶ Press the **Key management** soft key
> The control opens the **Certificate and keys** window.
▶ Select the key to be deleted
▶ Press the **Delete SSH key** soft key
> The control deletes the selected key.

Disabling non-secure connections in the firewall

The DNC protocols LSV2 and RPC should be blocked in the firewall to ensure that the use of secure connections provides a real IT security advantage for the control.

This requires that the following parties change to secure connections:

■ Machine manufacturers with all external applications (e.g., pick-and-place robots)

> **ⓘ** If the additional application is connected via **machine network X116**, there is no need to switch to an encrypted connection.

■ Users with custom external applications

If you have ensured that the connections of all parties are secure, the LSV2 and RPC DNC protocols can be blocked in the **Firewall**.

To block the notifications in the firewall:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Firewall** menu item

▶ Select the **Prohibit all** method for **DNC** and **LSV2**

▶ Select the **Apply** function

> The control saves your changes.

▶ Close the window with **OK**

Login to user administration

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

The login dialog provides the following options:

■ Users who logged in at least once

■ **Other** user
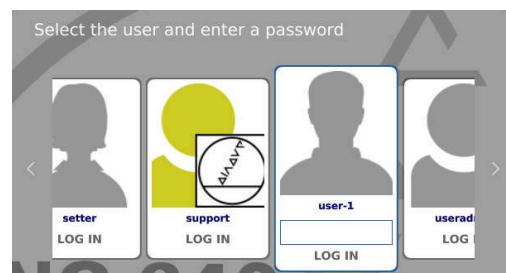
Logging in a user for the first time

To log in as a user for the first time, you need to use the **Other** input field.

To log in as a user with the **Other** option for 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 opens a window with the **Password expired** message. **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 user is shown in the login dialog.

Logging in a known user with password

To log in as a user who is displayed in the login dialog:

▶ Select the respective user in the login dialog

> The control enlarges the user icon you selected.

▶ Enter the user password

> The control uses the selected user to log you in.

> ℹ The control shows in the Login dialog whether CAPS LOCK is active.

Logging in a user with token

To log in as a user with a token:

▶ Hold the token close to the reader

▶ Enter PIN, if required

> The control uses the selected user to log you in.

▶ Remove the token from the reader

Password requirements

> **ⓘ** 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)

Keep in mind that an administrator can define the requirements to
be met by a password. The password requirements include:

- Minimum length
- Minimum number of different types of characters
  - Capital letters
  - Small letters
  - Numbers
  - Special characters
- Maximum length of character strings (e.g., 54321 = string of five
  characters)
- The number of characters must match the dictionary check
- Minimum number of characters changed compared to previous
  password

If the new password does not meet the requirements, an error
message will be issued. This means that you have to enter a
different password.

> **ⓘ** Administrators can specify the expiration of passwords. If
> you do not change your password within the valid period,
> you will no longer be able to log on as that user. In this
> case, an administrator needs to reset the user password
> before you can log on again.
> - ▶ Change your password at regular intervals
>
>   **Further information:** "Changing the current user's
>   password", Page 368
> - ▶ Take note of the password change warnings

Switching or logging out users

Use the **Switch-off** HEROS menu item or the icon of the same name at bottom right in the menu bar to open the **Switch off and restart** window.

The control provides the following options:

■ **Switch-off**:
- ■ The control stops and ends all additional programs and functions
- ■ The system is shut down
- ■ The control is switched off

■ **Restart**:
- ■ The control stops and ends all additional programs and functions
- ■ The system is restarted

■ **Log out**:
- ■ The control ends all additional programs
- ■ The user is logged out
- ■ The login window appears

> **i** To continue, you need to log in a new user and enter his password.
> The NC program continues running under the previously logged-on user.

■ **Switch user**:
- ■ The login window appears
- ■ The user is not logged out

> **i** You can close the login window with **Cancel** without entering a password.
> All additional programs and the NC programs started by the logged-on user continue running.
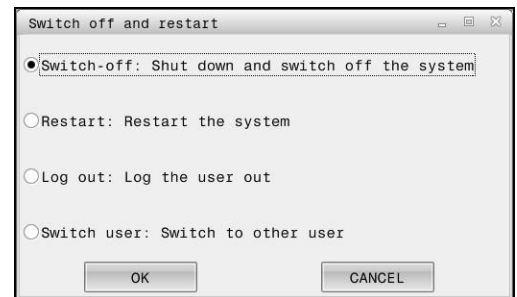
Screensaver with lockout

You can use the screensaver to lock the control. NC programs that have already been started continue running during this period.

> **i** You need to enter a password to unlock the screensaver again.
> **Further information:** "Login to user administration", Page 360
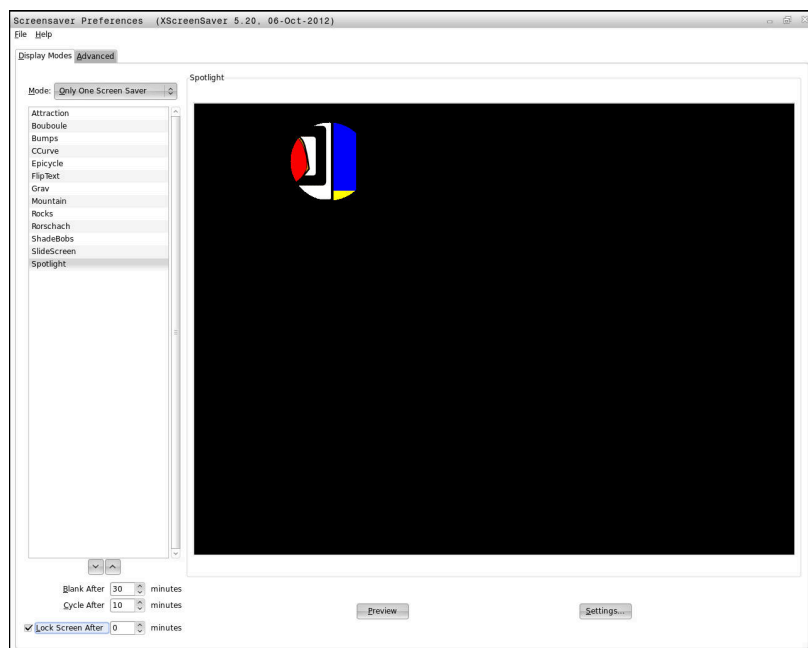
To access the screensaver settings in the **HEROS menu**:

▶ Press the **DIADUR** key to open the **HEROS menu**

▶ Select the **Settings** menu item

▶ Select the **Screensaver** menu item

The screensaver provides the following options:

■ Use the **Blank After** setting to define after how many minutes the screensaver should turn on.

■ Use the **Lock Screen After** setting to activate the lock with password protection.

■ The time value behind **Lock screen after** defines the delay after which the lock will take effect after the screensaver has been turned on. A value of **0** means that the lock becomes effective immediately after the screensaver has been turned on.

When the lock is active and you use one of the input devices (for example, by moving the mouse), the screensaver disappears. The control displays a lock screen instead.
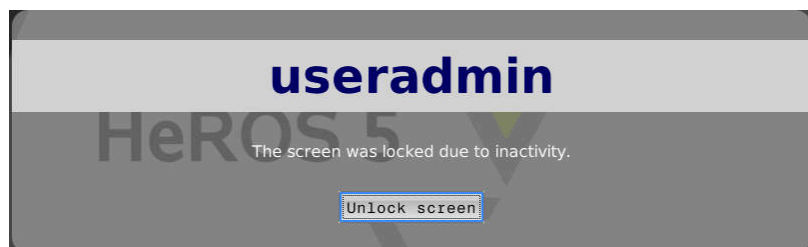
Use **Unlock screen** or **Enter** to open the login window again.

HOME directory

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by the respectively logged-in users as well as users with the HEROS.Admin role.
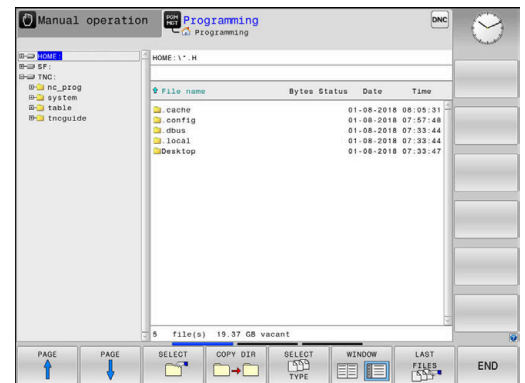
'public' directory

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.
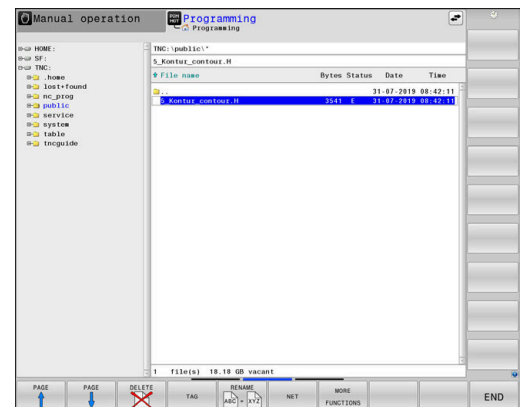
Setting advanced file access rights

In order to control the use of individual files in the **public** directory, HEIDENHAIN provides the **ADVANCED ACCESS RIGHTS** function that allows you to restrict access on a file level.

To call the **ADVANCED ACCESS RIGHTS** function:

▶ Select the **Programming** operating mode

▶ Press the **PGM-MGT** key

▶ Switch to horizontal soft-key row 2
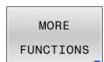
▶ Press the **MORE FUNCTIONS** soft key

▶ Switch to horizontal soft-key row 2

▶ Press the **ADVANCED ACCESS RIGHTS** soft key

> The control opens the **Set advanced access rights** window.

Setting file access rights

If you transfer files to the **public** directory or create files there, the control considers the logged-in user as the file owner. The owner can control access to his own files.

> **i** Access rights can only be defined for files located in the **public** directory.
>
> For all files stored on the **TNC:** drive instead of the **public** directory, the **user** function user will automatically be assigned as the owner.

You can specify the access rights for the following users:

- **Owner:**

  The person who owns the file

- **Group:**

  A selected Linux group or users with a specific HEIDENHAIN access right

- **Other:**

  All users not belonging to the previously selected Linux group or not having the specified HEIDENHAIN access right.

You can set access rights as follows:

- **Read**

  The file can be viewed

- **Write**

  The file can be edited

- **Run**

  The file can be executed

Using the soft keys in the **Set advanced access rights** window, you can select or deselect all access rights for the users:

TOGGLE ACCESS FOR USER	▶ Select or deselect all access rights for **Owner:**
TOGGLE ACCESS FOR GROUP	▶ Select or deselect all access rights for the **Group:**
TOGGLE ACCESS FOR OTHERS	▶ Select or deselect all access rights for **Other:**

Set advanced access rights

Name: 5_Kontur_contour.H

Owner: useradmin
  ☒ Read   ☒ Write   ☒ Run

Group: user
  ☒ Read   ☐ Write   ☒ Run

Other:
  ☒ Read   ☐ Write   ☒ Run

            OK   CANCEL

To select the access rights for a group:
- ▶ Call the **ADVANCED ACCESS RIGHTS** function
- ▶ Select the desired group in the selection menu
- ▶ Select or deselect the access rights as required
- ▷ Any changes made are highlighted in red.
- ▶ Press **OK**
- ▷ The changes made to access rights are applied.
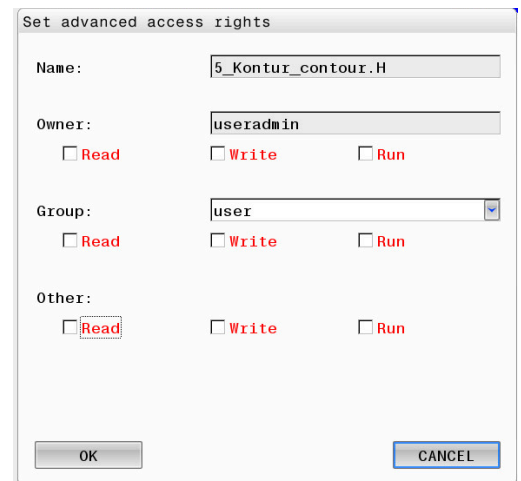
Current User

With **Current User**, you can view the group rights of the currently logged-on user in the **HEROS** menu.

> ℹ️ 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:** "HEIDENHAIN function users", Page 350
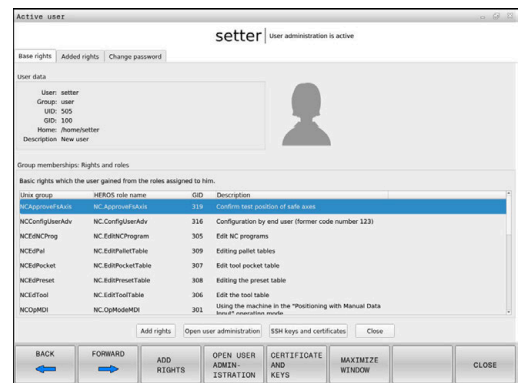
To call the **Current User**:
- ▶ Press the **DIADUR** key to open the **HEROS menu** menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Current User** menu icon

Temporarily changing the current user's rights

User administration allows you to temporarily increase the current user's rights by adding the rights of a user you selected.

To temporarily increase a user's rights:
- ▶ Call **Current User**
- ▶ Press the **Add rights** soft key
- ▶ Select the user
- ▶ Enter the user's user name
- ▶ Enter the selected user's password
- ▷ The control temporarily increases the logged-on user's rights by adding the rights of the user specified in **Add rights**.

> ℹ️ You can temporarily enable the rights of **oem** function users. For this purpose, enter the appropriate code number or the password defined by the machine manufacturer.

You have the following options to remove the rights that have temporarily been added:
- ▪ Enter the code number **0**
- ▪ User logout
- ▪ Press the **Remove added rights** soft key

To select the **Remove added rights** soft key:

▶ Call **Current User**

▶ Select the **Added rights** tab

▶ Press the **Remove added rights** soft key

Changing the current user's password

In the **Current User** menu item, you can change the current user's password.

To change the current user's password:

▶ Call **Current User**

▶ Select the **Change password** tab

▶ Enter the old password

▶ Press the **Verify old password** soft key

> The control checks whether you have correctly entered your old password.

> If the password is correct, the fields **New password** and **Repeat password** are enabled.

▶ Enter a new password

▶ Repeat the new password

▶ Press the **Set new password** soft key

> The control compares the requirements defined for passwords by the administrator to the password you entered.

   Further information: "Login to user administration", Page 360

> The message **Password was changed successfully** appears.

Defining login with token

The control also permits login with a token. This ensures secure login without the user having to enter a password.

⚙ Refer to your machine manual.

   The machine manufacturer must prepare the machine for use with a token. For example, an appropriate reader must be mounted on the machine.

In the **Current User** menu item, you can define login with a token for the current user.

To create a token:

▶ Call **Current User**

▶ Select **Create token**

▶ Select the type of token with **Switch type**, if required

▶ Enter the user's password

▶ Enter PIN, if required

▶ Hold the token close to the reader

▶ Select **Reload list**

▶ Choose a token from the list

▶ Select **Start writing**

▶ Enter PIN, if defined

> The control starts the write process.

▶ Hold the token close to the reader until the end of the write process

> When writing is complete, the control shows a message.

Delete token allows you to delete the created token and use password entry again.

Dialog for requesting additional rights

If you do not have the rights required for a specific **HEROS menu** item, the control opens the window for requesting additional rights:

In this window, you can temporarily obtain more rights by adding another user's rights.

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

i If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.

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.
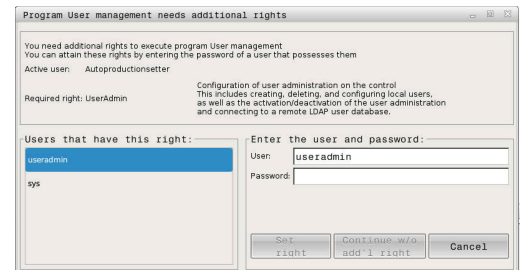
Obtaining more rights

To temporarily add another user's rights to your rights:

▶ Select the user whose rights you need

▶ Enter the user's name

▶ Enter the user's password

▶ Press the **Set right** soft key

> The control adds the entered user's rights to your rights.

Further information: "Current User", Page 367

9.8 Changing the HEROS conversational language

The HEROS conversational language is internally based on the NC conversational language. Therefore the **HEROS menu** and the control cannot permanently be set to two different conversational languages.

When you change the NC conversational language, the HEROS conversational language will only reflect this change after a restart of the control.

> **i** With the optional machine parameter **applyCfgLanguage** (no. 101305), you define the behavior of the control if the NC conversational language and in the HEROS conversational language do not match.

The following link provides information on how to change the NC conversational language:

Further information: "List of user parameters", Page 374

Changing the language layout of the keyboard

You can change the language layout of the keyboard for HEROS applications.

Proceed as follows to change the language layout of the keyboard for HEROS applications:

▶ Select the HEROS menu icon
▶ Select **Settings**
▶ Select **Language/Keyboards**
> The control opens the **helocale** window.
▶ Select the **Keyboards** tab
▶ Select the desired keyboard layout
▶ Select **Apply**
▶ Press **OK**
▶ Select **Apply**
> The changes are applied.

10

Tables and Overviews

10.1 Machine-specific user parameters

Application

The parameter values are entered in the **configuration editor**.

> ⚙ 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.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **Settings for screen displays**) that gives information about the parameters it contains.

Calling the configuration editor

Proceed as follows:

MOD	▶ Press the **MOD** key

↑	▶ Select the **Code-number entry** function, if necessary
	▶ Enter the code number **123**

ENT	▶ Confirm with the **ENT** key
	> The control displays the list of available parameters in the tree view.

Depiction of the parameters

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

- ▪   Branch exists but is closed

- ▪   Branch is open

- ▪   Empty object, cannot be opened

- ▪   Initialized machine parameter

- ▪   Uninitialized (optional) machine parameter

- ▪   Can be read but not edited

- ▪   Can neither be read nor edited

The folder icon indicates the object type:

- ▪   Key (group name)

- ▪   List

- ▪   Entity (parameter object)

> **ℹ** The icons of parameters and objects that are not yet active appear dimmed. These can be activated with the **MORE FUNCTIONS** and **INSERT** soft keys.

Changing the parameters

Proceed as follows:

- ▶ Find the desired parameter
- ▶ Edit the value

| END | ▶ Press the **END** soft key to close the configuration editor |

| STORE | ▶ Press the **SAVE** soft key to save the changes |

> **ℹ** The control saves a change list of the last 20 changes to the configuration data. To undo changes, select the corresponding line and press the **MORE FUNCTIONS** and **CANCEL CHANGE** soft keys.

Changing the display of the parameters

You can change the display of the existing parameters in the configuration editor for the machine parameter. By default, the parameters are displayed with short, explanatory texts.

To display the actual system names of the parameters:

▶ Press the **Screen layout** key

▶ Press the **SHOW SYSTEM NAME** soft key

Follow the same procedure to return to the standard display.

Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

The control displays other information (e.g. unit of measurement, initial value, selection list) in addition to the help text. If the selected machine parameter matches a parameter in the previous control model, the corresponding MP number is displayed.

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.

Parameter settings

DisplaySettings

    Settings for screen display

        Display sequence and rules for axes

            [0] to [3]: depends on the available axes

                Key name of an object in CfgAxis

                    Key name of the axis to be displayed

                Axis designation

                    Axis designation to be used in place of the key name

                Display rule for the axis

                    ShowAlways
                    IfKinem
                    IfKinemAxis
                    IfNotKinemAxis
                    Never

        Display sequence and rules for axes in the REF display

            [0] to [5]: depends on the available axes

                See display sequence and rules for axes

        Type of position display in the position window

            NOML
            ACTL
            REF ACTL
            REF NOML
            LAG
            ACTDST
            REFDST
            M118

        Type of position display in the status display

            NOML
            ACTL
            REF ACTL
            REF NOML
            LAG
            ACTDST
            REFDST
            M118

        Definition of decimal separator for position display

            . point
            , comma

        Display of the feed rate in the Manual Operation and El. Handwheel modes of operation

            at axis key: Display the feed rate only if axis-direction key is pressed

Parameter settings

> **always minimum: Always display the feed rate**

Display of spindle position in the position display

> **during closed loop: Display spindle position only if spindle is in position feedback control**
>
> **during closed loop and M5: Display spindle position if spindle is in position feedback control and with M5**
>
> **during closed loop or M5 or tapping: Display spindle position if spindle is in position feedback control, with M5 and in spindle jog operation**

Locking of PRESET MANAGEMENT soft key

> **TRUE: Access to the preset table is locked**
>
> **FALSE: The preset table can be accessed by soft key**

Font size for program display

> **FONT_APPLICATION_SMALL**
>
> **FONT_APPLICATION_MEDIUM**

Sequence of icons in the display

> [0] to [19]: depends on the activated options
>
> > **e.g., S_PULSE**

Settings for display behavior: depends on the machine manufacturer

> **Entered by the machine manufacturer**

Parameter settings

DisplaySettings
    Display step for the individual axes
        List of all available axes
            Display step for position display in mm or degrees
                0.1
                0.05
                0.01
                0.005
                0.001
                0.0005
                0.0001
                0.00005
                0.00001
                0.000005
                0.000001

            Display step for position display in inches
                0.005
                0.001
                0.0005
                0.0001
                0.00005
                0.00001
                0.000005
                0.000001

DisplaySettings
    Definition of the unit of measure valid for the display
        Unit of measure for display in the user interface
            metric: Use the metric system
            inch: Use the inch system

DisplaySettings
    Format of the NC programs and cycle display
        Program entry in HEIDENHAIN Klartext or in ISO
            HEIDENHAIN: Klartext language for entering the program in Positioning with Manual Data Input operating mode
            ISO: G codes for entering the program in Positioning with Manual Data Input operating mode

---

Parameter settings

DisplaySettings

    Definition of the NC and PLC conversational languages

        NC conversational language

            ENGLISH

            GERMAN

            CZECH

            FRENCH

            ITALIAN

            SPANISH

            PORTUGUESE

            SWEDISH

            DANISH

            FINNISH

            DUTCH

            POLISH

            HUNGARIAN

            RUSSIAN

            CHINESE

            CHINESE_TRAD

            SLOVENIAN

            KOREAN

            NORWEGIAN

            ROMANIAN

            SLOVAK

            TURKISH

        Load the language of the NC control

            FALSE: Upon startup of the control, the language of the HEROS operating system is used

            TRUE: Upon startup of the control, the language from the machine parameters is used

        PLC conversational language

            See NC conversational language

        PLC error message language

            See NC conversational language

        Language for online help

            See NC conversational language

DisplaySettings

    Behavior during control start-up

        Acknowledge the "Power interrupted" message

            TRUE: Start-up of the control is not continued until the message has been acknowledged

            FALSE: The Power interrupted message does not appear

---

Parameter settings

DisplaySettings

    Display mode for time of day

        Selection of display mode

            Analog

            Digital

            Logo

            Analog and logo

            Digital and logo

            Analog on logo

            Digital on logo

DisplaySettings

    Link row on/off

        Display settings for link row

            FALSE: Switch off the information line in the operating-mode line

            TRUE: Switch on the information line in the operating-mode line

DisplaySettings

    Settings for 3D simulation graphics

        Model type of the 3D simulation graphics

            3D: Model depiction for complex machining operations with undercuts (compute-intensive)

            2.5D: Model depiction for 3-axis operations

            No Model: Model depiction is deactivated

        Model quality of the 3D simulation graphics

            very high: High resolution; the block end points can be depicted

            high: High resolution

            medium: Medium resolution

            low: Low resolution

         Reset tool paths in new BLK form

            ON: With new BLK form in Test Run, the tool paths are reset

            OFF: With new BLK form in Test Run, the tool paths are not reset

        Write graphics journal files after restart

            OFF: Do not generate journal data

            ON: Generate journal data after a restart for diagnostic purposes

DisplaySettings

    Position display settings

        Position display with TOOL CALL DL

            As Tool Length: The programmed oversize DL is considered to be the change in tool length for display of the workpiece-based position

            As Workpiece Oversize: The programmed oversize DL is considered to be the workpiece oversize for display of the workpiece-based position

Parameter settings

DisplaySettings

Table editor setting

Behavior when deleting tools from the pocket table

DISABLED: Deletion of tool is not possible

WITH_WARNING: Tool deletion is possible, but must be confirmed

WITHOUT_WARNING: Tool deletion is possible without warning

Behavior when deleting a tool's index entries

ALWAYS_ALLOWED: Deletion of index entries is always possible

TOOL_RULES: The behavior depends on the setting of the parameter "Behavior when deleting tools from the pocket table"

Show the RESET COLUMN T soft key

TRUE:The soft key is displayed. All tools can be deleted from the tool memory by the user

FALSE: The soft key is not displayed

Parameter settings

ProbeSettings

    Configuration of the tool calibration

        TT140_1

            M function for spindle orientation

                −1: Spindle orientation directly by NC

                0: Function inactive

                1 to 999: Number of the M function for spindle orientation

            Probing routine

                MultiDirections: Probing from multiple directions

                SingleDirection: Probing from one direction

            Probing direction for tool radius measurement: Depends on the tool axis

                X_Positive, Y_Positive, X_Negative, Y_Negative, Z_Positive, Z_Negative

            Distance from lower edge of tool to upper edge of stylus

                0.001 to 99.9999 [mm]

            Rapid traverse in probing cycle

                10 to 300 000 [mm/min]

            Probing feed rate during tool measurement

                1 to 30 000 [mm/min]

            Calculation of the probing feed rate

                ConstantTolerance: Calculation of the probing feed rate with constant tolerance

                VariableTolerance: Calculation of the probing feed rate with variable tolerance

                ConstantFeed: Constant probing feed rate

            Speed determination method

                Automatic: Automatically determine the speed

                MinSpindleSpeed: Use the minimum spindle speed

            Maximum permissible surface speed of the tool edge (milling tool circumference)

                1 to 129 [m/min]

            Maximum permissible speed for tool measurement

                0 to 1000 [rpm]:

            First maximum permissible measurement error during tool measurement

                0.001 to 0.999 [mm]

            Second maximum permissible measurement error during tool measurement

                0.001 to 0.999 [mm]

            NC stop during tool check

                True: When the breakage tolerance is exceeded, the NC program is stopped

                False: The NC program is not stopped

Parameter settings

NC stop during tool measurement
True: When the breakage tolerance is exceeded, the NC program is stopped
False: The NC program is not stopped

Change the tool table during tool check and tool measurement
AdaptOnMeasure: The table is changed after tool measurement
AdaptOnBoth: The table is changed after tool check and tool measurement
AdaptNever: The table is not changed after tool check and tool measurement

ProbeSettings
Configuration of a round stylus
TT140_1
Coordinates of the stylus center
[0]: X coordinate of the stylus center with respect to the machine datum [mm]
[1]: Y coordinate of the stylus center with respect to the machine datum [mm]
[2]: Z coordinate of the stylus center with respect to the machine datum [mm]

Set-up clearance above the stylus for pre-positioning
0.001 to 99 999.9999 [mm]

Safety zone around the stylus for pre-positioning: Set-up clearance in the plane perpendicular to the tool axis
0.001 to 99 999.9999 [mm]

ProbeSettings
Configuration of a rectangular stylus
TT140_1
Coordinates of the stylus center
[0]: X coordinate of the stylus center with respect to the machine datum [mm]
[1]: Y coordinate of the stylus center with respect to the machine datum [mm]
[2]: Z coordinate of the stylus center with respect to the machine datum [mm]

Set-up clearance above the stylus for pre-positioning
0.001 to 99 999.9999 [mm]

Safety zone around the stylus for pre-positioning: Set-up clearance in the plane perpendicular to the tool axis
0.001 to 99 999.9999 [mm]

Parameter settings

ChannelSettings

    CH_NC

        Active kinematics

            Kinematics to be activated

                List of machine kinematics

            The kinematics to be activated during control start-up

                List of machine kinematics

        Specify the behavior of the NC program

            Reset the machining time when program starts

                True: Machining time is reset

                False: Machining time is not reset

            PLC signal for the number of the pending fixed cycle

                Depends on the machine manufacturer

            PLC signal for type of current cycle execution

                0: If no machining cycle is currently being performed

                1: During pre-positioning

                2: During actual machining

        Configuration of machining cycles

            Overlap factor for pocket milling: Path overlap for Cycle 4 POCKET MILLING

                0.001 to 1.414

            Traverse after machining a contour pocket

                PosBeforeMachining: Tool moves to the same position as before the cycle was machined

                ToolAxClearanceHeight: Position the tool axis to clearance height

            Display the **Spindle ?** error message if M3/M4 is not active

                On: Issue error message

                Off: No error message

            Display the **Enter depth as negative**

                On: Issue error message

                Off: No error message

            Behavior when moving to wall of slot in the cylinder surface

                LineNormal: Approach on a straight line

                CircleTangential: Approach

            M function for spindle orientation in machining cycles

                −1: Spindle orientation directly by NC

                0: Function inactive

                1 to 999: Number of the M function for spindle orientation

Parameter settings

Do not show **Plunging type is not possible** error message

On: Error message is not displayed

Off: Error message is displayed

Behavior of M7 and M8 with Cycles 202 and 204

TRUE: At the end of Cycles 202 and 204, the status of M7 and M8 is restored to that before the cycle call

FALSE: At the end of Cycles 202 and 204, the status of M7 and M8 is not restored automatically

Do not show the **Residual material** warning

Never: The warning is always displayed

NCOnly: The warning is suppressed during program run

Always: The warning is never displayed

Special spindle parameters

Potentiometer for feed rate during thread cutting

SpindlePotentiometer: During thread cutting, the potentiometer is effective for the spindle-speed override. The potentiometer for the feed-rate override is disabled

FeedPotentiometer: During thread cutting, the potentiometer is effective for the feed-rate override. The potentiometer for the spindle-speed override is not active

Waiting time at reversal point in thread base: The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation

−999999999 to 999999999 [s]

Advanced switching time of spindle: The spindle is stopped for this period of time before reaching the bottom of the thread.

−999999999 to 999999999 [s]

Limitation of spindle speed with Cycles 17, 207, and 18

TRUE: With small thread depths, the spindle speed is limited so that it runs at constant speed approx. 1/3 of the time

FALSE: No limitation of the spindle speed

Parameter settings

Settings for the NC editor

Generate backup files

TRUE: Generate backup file after editing NC programs

FALSE: Do not generate backup file after editing NC programs

Behavior of the cursor after deletion of lines

TRUE: Cursor is placed on the preceding line after deletion (behavior of iTNC)

FALSE: Cursor is placed on the following line after deletion

Behavior of the cursor on the first or last line

TRUE: Cursor jumps from end to beginning of program

FALSE: Cursor is not allowed to jump from end to beginning of program

Line break on blocks with more than one line

ALL: Always display all lines

ACT: Only display the lines of the active block completely

NO: Only display all lines when block is edited

Activate help graphics when entering cycle data

TRUE: Always display help graphics during input

FALSE: Only display help graphics when the CYCLE HELP soft key is set to ON. The CYCLE HELP OFF/ON soft key is displayed in Programming operating mode after pressing the "Screen layout" key

Behavior of the soft-key row after entering a cycle

TRUE: The cycle soft-key row remains active after a cycle definition

FALSE: The cycle soft-key row is hidden after a cycle definition

Confirmation request when deleting a block

TRUE: Display confirmation request when deleting an NC block

FALSE: Do not display confirmation request when deleting an NC block

Line number up to which a test of the NC program is to be run: Program length for which the geometry is to be checked

100 to 50000

ISO programming: Increments for creation of ISO blocks in the program

0 to 250

Specify programmable axes

TRUE: Use the specified axis configuration

FALSE: Use the default axis configuration XYZABCUVW

Line number up to which identical syntax elements are searched for: Search for selected elements with up or down cursor keys

500 to 50000

Hide FUNCTION PAARAXCOMP/PARAXMODE

FALSE: PARAXCOMP and PARAXMODE functions permitted

Parameter settings

> **TRUE: PARAXCOMP and PARAXMODE functions blocked**

File management settings

> Display of dependent files
>
> > **MANUAL: Dependent files are displayed**
> > **AUTOMATIC: Dependent files are not displayed**

Settings for tool usage file

> Timeout for creation of tool-usage files
>
> > **1 to 500 [min]**
>
> Create tool-usage file for NC program
>
> > **NotAutoCreate: Do not create a tool usage list when a program is selected**
> > **OnProgSelectionIfNotExist: When a program is selected, create a list if does not yet exist**
> > **OnProgSelectionIfNecessary: When a program is selected, create a list if it does not yet exist or is obsolete**
> > **OnProgSelectionAndModify: When a program is selected, create a list if it does not yet exist, is obsolete, or the program is changed**
>
> Create a pallet usage file
>
> > **NotAutoCreate: Do not create a tool usage list when a pallet is selected**
> > **OnProgSelectionIfNotExist: When a pallet is selected, a list is created if it does not yet exist**
> > **OnProgSelectionIfNecessary: When a pallet is selected, a list is created if it does not yet exist or is obsolete**
> > **OnProgSelectionAndModify: When a pallet is selected, a list is created if it does not yet exist, is obsolete, or the program is changed**

Paths for the end user

> List of drives or directories: These machine parameters are effective only for a Windows programming station
>
> > **Drives or directories entered here are shown in the control's file manager**
>
> FN 16 output path for execution
>
> > **Path for FN 16 output when no path is defined in the NC program**
>
> FN 16 output path for the Programming and Test Run operating modes
>
> > **Path for FN 16 output when no path is defined in the NC program**

Parameter settings

---

serialInterfaceRS232

    Data record belonging to the serial port

        Release of the RS-232-C/V.24 interface in the file manager

            TRUE: Drive RS232: It is displayed and you can copy files via this interface

            FALSE: The drive is not displayed

        Data transfer rate for LSV2 communication in baud

            BAUD_110

            BAUD_150

            BAUD_300

            BAUD_600

            BAUD_1200

            BAUD_2400

            BAUD_4800

            BAUD_9600

            BAUD_19200

            BAUD_38400

            BAUD_57600

            BAUD_115200

    Definition of data blocks for the serial ports

        RS-232

            Data transfer rate in baud

                BAUD_110

                BAUD_150

                BAUD_300

                BAUD_600

                BAUD_1200

                BAUD_2400

                BAUD_4800

                BAUD_9600

                BAUD_19200

                BAUD_38400

                BAUD_57600

                BAUD_115200

            Data transfer protocol

                STANDARD: Standard data transfer. Data transferred line-by-line

                BLOCKWISE: Packet-based data transfer, ACK/NAK protocol. The control characters ACK (Acknowledge) and NAK (not Acknowledge) are used to control blockwise data transfer.

                RAW_DATA: Data transferred without protocol. Transfer of characters without control characters. Protocol intended for transfer of data of the PLC.

            Data bits in each transferred character

                7 bits: 7 data bits are transferred for each character transferred

---

Parameter settings

> **8 bits: 8 data bits are transferred for each character transferred**

Type of parity checking
> **NONE: No parity**
>
> **EVEN: Even parity**
>
> **ODD: Odd parity**

Number of stop bits
> **1 stop bit: 1 stop bit is appended after each transferred character.**
>
> **2 stop bits: 2 stop bits are appended after each transferred character.**

Specify type of handshake:
> **NONE: No data-flow check. Handshake not active**
>
> **RTS_CTS: Hardware handshaking; transfer stop is active through RTS**
>
> **XON_XOFF: Software handshake; transfer stop by DC3 (XOFF) active**

File system for file operation via serial interface
> **EXT: Minimum file system to be used for non-HEIDENHAIN devices, such as printers, punches, or PCs with other data transfer software**
>
> **FE1: Use this setting for communication with the external HEIDENHAIN Floppy Disk Unit FE 4xx or for communication with the HEIDENHAIN PC software TNCserver.**

Block Check Character (BCC) is not a control character
> **TRUE: Ensures that the check sum does not correspond to a control character**
>
> **FALSE: Function not active**

Status of the RTS line
> **TRUE: The idle state of the RTS line is logical LOW**
>
> **FALSE: The idle state of the RTS line is logical HIGH**

Define behavior after receipt of ETX
> **TRUE: No EOT control character is sent after reception of an ETX control character.**
>
> **FALSE: The control sends an EOT control character after reception of an ETX control character.**

Further information: "Serial interfaces on the TNC 128", Page 305

Further information: User's Manual for **Setup, Testing and Running NC Programs**

10.2 Pin layout and cables for data interfaces

V.24/RS-232-C interface for HEIDENHAIN devices

> ℹ 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

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

10.3 Specifications

Explanation of symbols

- ■ Standard function
- □ Axis option
- **1** Advanced Function Set 1
- **x** Software option, except Advanced Function Set 1 and Advanced Function Set 2

Specifications		
Components	■	Main computer
	■	Operating panel
	■	Screen with soft keys
Program memory	■	2 GB
Input resolution and display step	■	As fine as 0.1 μm for linear axes
	■	As fine as 0.000 1° for rotary axes
Input range	■	Maximum 999 999 999 mm or 999 999 999°
Block processing time	■	6 ms
Axis feedback control	■	Position-loop resolution: signal period of the position encoder/4096
	■	Position controller cycle time: 200 μs (100 μs with option 49)
	■	Speed controller cycle time: 200 μs (100 μs with option 49)
	■	Current controller cycle time: minimum 100 μs (minimum 50 μs with option 49)
Spindle speed	■	max. 100 000 rpm (with 2 pole pairs)
Error compensation	■	Linear and nonlinear axis errors, backlash, thermal expansion
	■	Static friction, sliding friction

Specifications

Data interfaces	■	One each: RS-232-C/V.24 max. 115 kbit/s
	■	Extended data interface with LSV-2 protocol for remote operation of the control through the data interface with the TNCremo or TNCremoPlus software
	■	2 x Gigabit Ethernet interface 1000BASE-T
	■	3 USB ports: 1 front USB 2.0 port and 2 rear USB 3.0 ports)
	x	HEIDENHAIN-DNC for communication between a Windows application and TNC (DCOM interface)
	x	OPC UA NC Server
		Stable and reliable interface for the connection of leading-edge industrial applications
Ambient temperature	■	Operation: +5 °C to +45 °C
	■	Storage: -20 °C to +60 °C

Input formats and units of control functions

Positions, coordinates, chamfer lengths	−99 999.9999 to +99 999.9999 (5, 4: number of digits before and after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	32 characters, enclosed by quotation marks in **TOOL CALL** block. Permitted special characters: # $ % & . , - _
Detail values for tool compensation	−99.9999 to +99.9999 (2, 4) [mm]
Spindle speeds	0 to 99 999.999 (5, 3) [rpm]
Feed rates	0 to 99 999.999 (5, 3) [mm/min] or [mm/tooth] or [mm/1]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	−99.9999 to +99.9999 (2, 4) [mm]
Angle for spindle orientation	0 to 360.0000 (3, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 9999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	−999 999 999.999999 to +999 999 999.999999 (9, 6)
Labels (LBL) for program jumps	0 to 65535 (5, 0)
Labels (LBL) for program jumps	Any text string in quotes ("")
Number of program-section repeats REP	1 to 65 534 (5, 0)
Error number for Q parameter function FN 14	0 to 1199 (4, 0)

User functions

User functions	Standard	Option	Meaning
Short description	✓		Basic version: 3 axes plus closed-loop spindle
		0	1st additional axis for 4 axes plus closed-loop spindle
		1	2nd additional axis for 5 axes plus closed-loop spindle
Program entry			In HEIDENHAIN Klartext format
Position entry	✓		Nominal positions for straight lines in Cartesian coordinates
	✓		Incremental or absolute dimensions
	✓		Display and entry in mm or inches
Tool tables	✓		Multiple tool tables with any number of tools
Cutting data	✓		Automatic calculation of spindle speed, cutting speed, feed per tooth, and feed per revolution
Program jumps	✓		Subprograms
	✓		Program section repeats
	✓		External NC programs
Machining cycles	✓		Cycles for drilling, and conventional and rigid tapping
		19	Cycles for pecking, reaming, boring, and counterboring
	✓		Roughing and finishing rectangular pockets
	✓		Roughing and finishing rectangular studs
	✓		Cycles for clearing level surfaces
	✓		Face milling
	✓		Cartesian and polar point patterns
	✓		OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	✓		Datum shift, mirroring
	✓		Scaling factor (axis-specific)
Q parameters Programming with variables	✓		Mathematical functions =, +, −, *, /, roots
	✓		Logical operations (=, ≠, <, >)
	✓		Calculating with parentheses
	✓		$\sin \alpha$, $\cos \alpha$, $\tan \alpha$, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	✓		Functions for calculation of circles
	✓		String parameters

User functions	Standard	Option	Meaning
Programming aids	✓		Calculator
	✓		Color highlighting of syntax elements
	✓		Complete list of all current error messages
	✓		Context-sensitive help function
	✓		Graphic support for the programming of cycles
	✓		Comment and structure blocks in the NC program
Teach-In	✓		Actual positions can be transferred directly to the NC program
Test graphics Display modes	✓		Graphic simulation before a program run, even while another NC program is being run
	✓		Plan view / projection in 3 planes / 3D view
	✓		Detail enlargement
Programming graphics	✓		In the **Programming** operating mode, the contours of the NC blocks are drawn on screen while they are being entered (2D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	✓		Graphic simulation of real-time machining in plan view / projection in 3 planes / 3D view
Machining time	✓		Calculation of machining time in the **Test Run** operating mode
	✓		Display of the current machining time in the **Program Run, Single Block** and **Program Run, Full Sequence** operating modes
Preset management	✓		For saving any datums
Returning to the contour	✓		Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining
	✓		NC program interruption, contour departure and return
Datum tables	✓		Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	✓		Calibrating the touch probe
	✓		Manual presetting
	✓		Tools can be measured automatically

> **ℹ** For a detailed overview of the user functions, see the brochure for the TNC 128. You can find the brochures related to the product range of CNC controls in the download area of the HEIDENHAIN website.

Accessories

Accessories		
Electronic handwheels	■	HR 510: Portable handwheel
	■	HR 550 FS: Portable radio handwheel with display
	■	HR 520: Portable handwheel with display
	■	HR 130: Panel-mounted handwheel
	■	HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
Touch probes	■	TS 248: workpiece touch trigger probe with cable connection
	■	TS 260: workpiece touch trigger probe with cable connection
	■	TT 160: tool touch trigger probe
	■	KT 130: Simple touch trigger probe with cable connection

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☎ +49 8669 31-3104
  service.ms-support@heidenhain.de
NC support ☎ +49 8669 31-3101
  service.nc-support@heidenhain.de
NC programming ☎ +49 8669 31-3103
  service.nc-pgm@heidenhain.de
PLC programming ☎ +49 8669 31-3102
  service.plc@heidenhain.de
APP programming ☎ +49 8669 31-3106
  service.app@heidenhain.de

www.heidenhain.com

Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.

For more details on touch probes and vision systems:
www.heidenhain.com/products/touch-probes-and-vision-systems