



HEIDENHAIN



TNC 128

User's Manual
Setup, Testing and Running
NC Programs







NC Software
771841-07

English (en)
10/2018






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
 ... 	Select the coordinate axes or enter them in the NC program
 ... 	Numbers
 	Decimal separator / Reverse algebraic sign
 	Polar coordinate entry / Incremental values
	Q parameter programming / Q parameter status
	Capture actual position
	Skip dialog questions, delete words
	Confirm entry and resume dialog
	Conclude the NC block, end your input
	Clear entries or error message
	Abort dialog, delete program section











Tool functions

Key	Function
	Define tool data in the NC program
	Call tool data





Managing NC programs and files, control functions

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

Navigation keys

Key	Function
 	Position the cursor
	Go directly to NC blocks, cycles, and parameter functions
	Navigate to the beginning of a program or table
	Navigate to the end of the program or table row
	Navigate up one page
	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
 	Define and call cycles
 	Enter and call subprograms and program section repeats

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Contents

1 Fundamentals.....21

2 First steps.....33

3 Fundamentals.....45

4 Tools.....97

5 Setup.....129

6 Testing and running.....181

7 Special Functions.....227

8 MOD functions.....231

9 HEROS functions.....253

10 Tables and Overviews.....321

1	Fundamentals.....	21
1.1	About this manual.....	22
1.2	Control model, software and features.....	24
	Software options.....	25
	New functions 77184x-06.....	27
	New functions 77184x-07.....	29

2	First steps.....	33
2.1	Overview.....	34
2.2	Switching on the machine.....	35
	Acknowledging the power interruption and moving to the reference points.....	35
2.3	Graphically testing the workpiece.....	36
	Selecting the Test Run operating mode.....	36
	Selecting the tool table.....	36
	Selecting the NC program.....	37
	Selecting the screen layout and the view.....	37
	Starting the test run.....	38
2.4	Setting up tools.....	39
	Selecting the Manual operation mode.....	39
	Preparing and measuring tools.....	39
	Editing the TOOL.T tool table.....	40
	Editing the TOOL_PTCH pocket table.....	41
2.5	Workpiece setup.....	42
	Selecting the correct operating mode.....	42
	Clamping the workpiece.....	42
	Presetting with a 3-D touch probe (option number 17).....	43
2.6	Machining the workpiece.....	44
	Selecting the Program run, single block or Program run, full sequence operating mode.....	44
	Selecting the NC program.....	44
	Starting an NC program.....	44

3	Fundamentals.....	45
3.1	The TNC 128.....	46
	HEIDENHAIN Klartext.....	46
	Compatibility.....	46
	Data security and data protection.....	47
3.2	Visual display unit and operating panel.....	49
	Display screen.....	49
	Setting the screen layout.....	49
	Control panel.....	50
	Screen keypad.....	50
3.3	Modes of operation.....	52
	Manual Operation and El. Handwheel.....	52
	Positioning with Manual Data Input.....	52
	Programming.....	52
	Test Run.....	53
	Program Run, Full Sequence and Program Run, Single Block.....	53
3.4	Status displays.....	54
	General status display.....	54
	Additional status displays.....	55
3.5	File management.....	60
	Files.....	60
	Displaying externally generated files on the control.....	62
	Directories.....	62
	Paths.....	62
	Calling the file manager.....	63
	Additional functions.....	64
	Selecting drives, directories and files.....	65
	Choosing one of the last files selected.....	67
	USB devices on the control.....	68
	Data transfer to or from an external data carrier.....	69
	The control in a network.....	70
	Data backup.....	72
	Importing a file from an iTNC 530.....	72
	Additional tools for management of external file types.....	73
3.6	Error messages and help system.....	82
	Error messages.....	82
	TNCguide context-sensitive help system.....	87
3.7	NC fundamentals.....	93
	Position encoders and reference marks.....	93

3.8	Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels.....	94
	3-D touch probes.....	94
	HR electronic handwheels.....	95

4	Tools.....	97
4.1	Tool data.....	98
	Tool number, tool name.....	98
	Tool length L.....	98
	Tool radius R.....	98
	Tool table fundamentals.....	99
	Entering tool data into the table.....	103
	Importing tool tables.....	108
	Overwriting tool data from an external PC.....	109
	Pocket table for tool changer.....	110
	Tool change.....	113
	Tool usage test.....	114
4.2	Tool management.....	117
	Basics.....	117
	Calling tool management.....	118
	Editing tool management.....	119
4.3	Tool carrier management.....	123
	Fundamentals.....	123
	Saving tool carrier templates.....	123
	Assigning input parameters to tool carriers.....	124
	Allocating parameterized tool carriers.....	127

5	Setup.....	129
5.1	Switch-on, switch-off.....	130
	Switch-on.....	130
	Traversing reference points.....	132
	Switch-off.....	133
5.2	Moving the machine axes.....	134
	Note.....	134
	Moving the axis with the axis direction keys.....	134
	Incremental jog positioning.....	135
	Traverse with the HR 510 electronic handwheel.....	136
	Moving with the electronic display handwheels.....	137
5.3	Spindle speed S, feed rate F and miscellaneous function M.....	148
	Application.....	148
	Entering values.....	148
	Changing the spindle speed and feed rate.....	149
	Feed rate limit F MAX.....	149
5.4	Preset management.....	151
	Note.....	151
	Saving presets in the table.....	151
	Protecting presets from being overwritten.....	155
	Activating a preset.....	157
5.5	Presetting without a 3-D touch probe.....	158
	Note.....	158
	Preparation.....	158
	Presetting with an end mill.....	159
	Using touch probe functions with mechanical probes or measuring dials.....	160
5.6	Using a 3-D touch probe (option 17).....	161
	Overview.....	161
	Suppressing touch probe monitoring.....	163
	Functions in touch probe cycles.....	163
	Selecting the probing cycle.....	165
	Recording measured values from the touch probe cycles.....	166
	Writing measured values from the touch probe cycles to a datum table.....	166
	Writing measured values from the touch probe cycles to the preset table.....	167
5.7	Calibrating 3-D touch probes (option 17).....	168
	Introduction.....	168
	Calibrating the effective length.....	169
	Calibrating the effective radius and compensating center misalignment.....	170
	Displaying calibration values.....	173

5.8	Presetting with a 3-D touch probe (option number 17)	174
	Overview	174
	Presetting on any axis	174
	Circle center as preset	175
	Setting a center line as preset	178
	Measuring workpieces with a 3-D touch probe	179

6	Testing and running.....	181
6.1	Graphics.....	182
	Application.....	182
	VIEW OPTIONS.....	183
	Tool.....	184
	View.....	185
	Rotating, zooming and moving a graphic.....	186
	Speed of the setting test runs.....	187
	Repeating graphic simulation.....	188
	Shifting the sectional plane.....	188
6.2	Showing the workpiece blank in the working space.....	189
	Application.....	189
6.3	Measuring.....	190
	Application.....	190
6.4	Optional program-run interruption.....	191
	Application.....	191
6.5	Skipping NC blocks.....	192
	Test Run and Program Run.....	192
	Positioning w/ Manual Data Input.....	193
6.6	Test run.....	194
	Application.....	194
	Test run execution.....	195
	Executing a Test Run up to a certain NC block.....	196
	GOTO function.....	197
	Display of NC programs.....	198
6.7	Program run.....	199
	Application.....	199
	Running an NC program.....	200
	Structuring NC programs.....	201
	Checking and changing Q parameters.....	202
	Interrupting, stopping or aborting machining.....	204
	Moving the machine axes during an interruption.....	207
	Resuming program run after an interruption.....	208
	Retraction after a power interruption.....	209
	Any entry into the NC program: Block scan.....	212
	Returning to the contour.....	217
6.8	Functions for program display.....	218
	Overview.....	218

6.9	Positioning w/ Manual Data Input operating mode.....	219
	Positioning with manual data input (MDI).....	220
	Saving NC programs from \$MDI.....	222
6.10	Entering miscellaneous functions M.....	223
	Fundamentals.....	223
6.11	Miscellaneous functions for program run inspection, spindle and coolant.....	224
	Overview.....	224
6.12	Miscellaneous functions for coordinate entries.....	225
	Programming machine-referenced coordinates: M91/M92.....	225

7	Special Functions.....	227
7.1	Defining a counter.....	228
	Application.....	228
	Define FUNCTION COUNT.....	229

8	MOD functions.....	231
8.1	MOD function.....	232
	Selecting MOD functions.....	232
	Changing the settings.....	232
	Exiting MOD functions.....	232
	Overview of MOD functions.....	233
8.2	Display software numbers.....	234
	Application.....	234
8.3	Entering the code number.....	234
	Application.....	234
	Functions provided to the machine tool builder by the code number dialog.....	235
8.4	Loading the machine configuration.....	236
	Application.....	236
8.5	Selecting the position display.....	237
	Application.....	237
8.6	Setting the unit of measure.....	239
	Application.....	239
8.7	Graphic settings.....	240
8.8	Setting the counter.....	241
8.9	Changing the machine settings.....	242
	Selecting the kinematics.....	242
	Entering traverse limits.....	243
	Generating a tool usage file.....	244
	Permitting or restricting external access.....	245
8.10	Configuring the HR 550FS radio handwheel.....	248
	Application.....	248
	Assigning the handwheel to a specific handwheel holder.....	248
	Setting the transmission channel.....	249
	Selecting the transmitter power.....	249
	Statistical data.....	250
8.11	Changing the system settings.....	251
	Set the system time.....	251
8.12	Displaying operating times.....	251
	Application.....	251

9	HEROS functions.....	253
9.1	Window manager.....	254
	Overview of taskbar.....	255
	Portscan.....	258
	Remote Service.....	259
	Printer.....	261
	SELinux security software.....	263
	State Reporting Interface (option 137).....	264
	VNC.....	267
	Backup and restore.....	270
9.2	Firewall.....	273
	Application.....	273
9.3	Setting up data interfaces.....	276
	Serial interfaces on the TNC 128.....	276
	Application.....	276
	Setting the RS-232 interface.....	276
	Set BAUD RATE (baud rate no. 106701).....	276
	Set protocol (protocol no. 106702).....	277
	Set data bits (dataBits no. 106703).....	277
	Check parity (parity no. 106704).....	277
	Set stop bits (stopBits no. 106705).....	277
	Set handshake (flowControl no. 106706).....	278
	File system for file operation (fileSystem no. 106707).....	278
	Block check character (bccAvoidCtrlChar no. 106708).....	278
	Condition of RTS line (rtsLow no. 106709).....	278
	Define behavior after receipt of ETX (noEotAfterEtx no. 106710).....	279
	Settings for the transmission of data using PC software TNCserver.....	279
	Setting the operating mode of the external device (fileSystem).....	279
	Software for data transfer.....	280
9.4	Ethernet interface.....	282
	Introduction.....	282
	Connection possibilities.....	282
	General network settings.....	283
	Device-specific network settings.....	289
9.5	SELinux security software.....	291
9.6	User administration.....	292
	Configuring the user administration.....	293
	Local LDAP database.....	295
	LDAP on remote computer.....	296
	Connection to a Windows domain.....	297
	Creating further users.....	300

Access rights.....303

HEIDENHAIN function users..... 304

Definition of roles..... 305

Rights..... 309

DNC connection with user authentication.....310

Login to user administration..... 314

Switching or logging out users..... 315

Screensaver with lockout.....316

HOME directory..... 317

Current User.....318

Dialog for requesting additional rights..... 319

9.7 Changing the HEROS conversational language.....320

10	Tables and Overviews.....	321
10.1	Machine-specific user parameters.....	322
	Application.....	322
	List of user parameter.....	325
10.2	Pin layout and cables for data interfaces.....	336
	RS-232-C/V.24 interface for HEIDENHAIN devices.....	336
	Non-HEIDENHAIN devices.....	338
	Ethernet interface RJ45 socket.....	338
10.3	Specifications.....	339
	User functions.....	341
	Accessories.....	343

1

Fundamentals

1.1 About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine tool builder'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 things other than personal injury, such as property damage**.

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.

In these instructions, you will find the following informational notes:



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



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



The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder 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

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.

Control model	NC software number
TNC 128	771841-07
TNC 128 Programming Station	771845-07

The machine manufacturer adapts the usable features of the control to his machine by the setting 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 3-D 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 that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Additional Axis (option 0 and option 1)	
Additional axis	Additional control loops 1 and 2
Touch Probe Functions (option 17)	
Touch probe functions	<div>Touch probe cycles:<ul style="list-style-type: none">■ Presetting in the Manual operation mode■ Tools can be measured automatically</div>
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
State Reporting Interface – SRI (option 137)	
HTTP accesses to the control status	<ul style="list-style-type: none">■ Reading out the times of status changes■ Reading out the active NC programs

Feature Content Level (upgrade functions)

Along with software options, significant further improvements of the control software are managed via the **Feature Content Level** upgrade functions. If you install a software update on your control you do not automatically have the functions available as covered by the FCL.



All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual as **FCL n**. The **n** signifies the serial number of the development status.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

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

This product uses open-source software. Further information is available on the control as follows:

- ▶ Press the **MOD** key
- ▶ Select **Code-number entry**
- ▶ **LICENSE INFO** soft key

New functions 77184x-06

- New **FUNCTION COUNT** function for controlling a counter.
- It is possible to comment out NC blocks.
- When multiple instances of the CAD viewer are open, they are shown somewhat smaller on the third desktop.
- With FN 16: F-PRINT, it is possible to enter references to Q parameters or QS parameters as the source and target.
- The FN 18 functions have been expanded.
- You can also open the tool-carrier files in the file management, see "Tool carrier management", Page 123
- With the **ADAPT NC PGM / TABLE** function, you can also import and modify freely definable tables, see "Importing tool tables", Page 108
- The machine tool builder can define update rules that make it possible, for example, to automatically remove umlauts from tables and NC programs when importing a table, see "Importing tool tables", Page 108
- A quick search for the tool name is possible in the tool table, see "Entering tool data into the table", Page 103
- The machine tool builder can disable the setting of presets in individual axes, see "Saving presets in the table", Page 151, see "Presetting with a 3-D touch probe (option number 17)", Page 174
- Line 0 of the preset table can also be edited manually, see "Saving presets in the table", Page 151
- The nodes in all tree structures can be expanded and collapsed by double-clicking them.
- New icon in the status display for mirrored machining, see "General status display", Page 54
- Graphic settings in the **Test Run** operating mode are permanently stored.
- In the **Test Run** operating mode, you can now choose between various ranges of traverse, see "Application", Page 189
- With the **TCH PROBE MONITOR OFF** soft key you can suppress touch-probe monitoring for 30 seconds, see "Suppressing touch probe monitoring", Page 163
- If the function for orienting the touch probe to the programmed probe direction is active, the number of spindle revolutions is limited when the guard door is open. In some cases, the direction of spindle rotation will change so that positioning will not always follow the shortest path.
- New machine parameter **iconPrioList** (no. 100813) for defining the order of icons in the status display, see "Machine-specific user parameters", Page 322
- The machine parameter **clearPathAtBlk** (no. 124203) enables you to specify whether the tool paths will be cleared with a new BLK FORM in the **Test Run** operating mode, see "Machine-specific user parameters", Page 322

Modified functions 77184x-06

- If you use locked tools, the control displays a warning in the **Programming** operating mode.
- The **TRANS DATUM AXIS** NC syntax can also be used within a contour in the SL cycle.
- Holes and threads are shown in light blue in the programming graphics.
- The sort order and the column widths are retained in the tool selection window when the control is switched off.
- If a file to be deleted does not exist, **FILE DELETE** no longer generates an error message.
- If a subprogram called with CALL PGM ends with **M2** or **M30**, the control issues a warning. The control automatically clears the warning as soon as you select another NC program.
- The time needed to paste a large amount of data into an NC program was considerably reduced.
- When you double-click a selection field of the table editor with the mouse or press the **ENT** key, a pop-up window opens.
- If you use locked tools, the control displays a warning in the **Test Run** operating mode, see "Test run", Page 194
- The control provides a positioning logic for returning to the contour, see "Returning to the contour", Page 217
- The positioning logic for returning to the contour with a replacement tool has changed, see "Tool change", Page 113
- If the control finds a stored interruption point on restart, you can resume the machining operation from that point, see "Any entry into the NC program: Block scan", Page 212
- The tool is shown in red in the graphics while it is in contact with the workpiece, and blue during air cuts, see "Tool", Page 184
- The positions of the sectional planes are no longer reset when a program or a new blank form is selected, see "Shifting the sectional plane", Page 188
- Spindle speeds can be entered with decimal places also in the **Manual operation** mode. The control displays the decimal places when the spindle speed is < 1000, see "Entering values", Page 148
- The control displays an error message in the header until it is cleared or replaced by a higher-priority error, see "Display of errors", Page 82
- To connect a USB stick you no longer have to press a soft key, see "Connecting and removing USB storage devices", Page 64
- The speed of setting the jog increment, spindle speed and feed rate was adjusted for electronic handwheels.
- The control automatically recognizes whether a table is to be imported or the table format is to be adapted, see "Importing tool tables", Page 108
- When configuration subfiles are modified, the control no longer aborts the test run, but only displays a warning.
- You can neither set nor modify a preset without having referenced the axes, see "Traversing reference points", Page 132

- The control issues a warning if the handwheel potentiometers are still active when the handwheel is deactivated, see "Moving with the electronic display handwheels", Page 137
- When using the HR 550 or HR 550FS handwheel, a warning is issued if the battery voltage is too low, see "Moving with the electronic display handwheels", Page 137
- The machine tool builder can define whether the **R-OFFS** offset will be taken into account for a tool with **CUT 0**, see "Tool data for automatic tool measurement", Page 105
- The machine tool builder can change the simulated tool change position, see "Test run", Page 194
- In the machine parameter **decimalCharakter** (no. 100805) you can define whether a period or a comma will be used as the decimal separator, see "Machine-specific user parameters", Page 322

New and modified cycle functions 77184x-06

- The parameters Q215, Q385, Q369 and Q386 were added to Cycles 256 **RECTANGULAR STUD**.
- Changes of details in Cycle 233: Monitors the tooth length (**LCUTS**) during finishing, increases the area by Q357 in the milling direction when roughing with milling strategies 0 to 3 (provided that no limit has been set in the milling direction).
- The technologically outdated Cycles 1, 2, 3, 4, 5, 17, 212, 213, 214, 215, 210, 211, 230, and 231 grouped under **OLD CYCLES** can no longer be inserted using the editor. These cycles can still be executed and edited, however.
- The tool touch probe cycles, such as Cycles 480, 481 and 482, can be hidden.
- New SERIAL column in the touch probe table.

New functions 77184x-07

- It is now possible to work with cutting data tables.
- In **Test Run** operating mode, a counter defined in the NC program is simulated.
- An NC program you called can be edited when it has been completely executed in the calling NC program.
- With **TOOL DEF**, you can use QS parameters for entering the data.
- You can now use QS parameters to read from and write to freely definable tables.
- The FN16 function was expanded to include the * input character that can be used to write comment lines.
- New output format for the FN16 function **%RS** that you can use to output texts without formatting.
- The FN 18 functions have been expanded.
- The new user administration enables you to create and administrate users with different access rights, see "User administration", Page 292
- With the new HOST COMPUTER MODE function, you can turn command over to an external host computer, see "Permitting or restricting external access", Page 245

- With the **State Reporting Interface (SRI)**, HEIDENHAIN provides a simple and reliable interface for acquiring the operating states of your machine, see "State Reporting Interface (option 137)", Page 264
- The screen layout soft keys were adapted, see "Modes of operation", Page 52
- The control checks all NC programs for completeness before machining. If you attempt to start an incomplete NC program, the control aborts with an error message, see "Data transfer to or from an external data carrier", Page 69.
- In the **Positioning w/ Manual Data Input** operating mode, you can now skip NC blocks, see "Skipping NC blocks", Page 192
- The appearance of the **Optional program run stop** has changed, see "Optional program-run interruption", Page 191
- You can use the key between **PGM MGT** and **ERR** to toggle between screens.
- The control supports USB devices with the exFAT file system, see "USB devices on the control", Page 68
- If the feed rate is less than 10, the control also shows one of the decimal place that have been entered, see "Entering values", Page 148
- In **Test Run** operating mode, the machine tool builder can define whether the tool table or the expanded tool management is opened.
- The machine tool builder defines which file types you will be able to import when using the **ADAPT NC PGM / TABLE** function, see "Importing a file from an iTNC 530", Page 72
- New machine parameter **CfgProgramCheck** (no. 129800) for defining settings for the tool usage files, see "List of user parameter", Page 325

Modified functions 77184x-07

- The cutting data calculator has been improved.
- The control does not run a tool change macro if neither a tool name nor a tool number is programmed in the tool call, but the same tool axis as in the previous **TOOL CALL** block.
- When using **SQL UPDATE** and **SQL INSERT**, the control checks the length of the table columns to be written to.
- When using the FN16 function, M_CLOSE and M_TRUNCATE have the same effect as far as output to the screen is concerned.
- In the **Test Run** operating mode, the **GOTO** key now has the same effect as in the other operating modes, see "GOTO function", Page 197
- The **ACTIVATE PRESET** soft key also updates the values of a line activated in the preset management, see "Activating a preset", Page 157
- From the third desktop you can switch to any operating mode using the operating mode keys.
- The additional status display in the **Test Run** operating mode was adapted to match that of the **Manual operation** mode, see "Additional status displays", Page 55

- The control allows updating of the web browser, see "Additional tools for management of external file types", Page 73
- The screensaver glideshow was removed.
- The machine tool builder can specify which M functions to allow in the **Manual Operation** mode, see "Application", Page 148
- The machine tool builder can define the default values for the L-OFFS and R-OFFS columns in the tool table, see "Entering tool data into the table", Page 103

New and modified cycle functions 77184x-07

- The REACTION column was added to the touch probe table.

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 Conversational 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

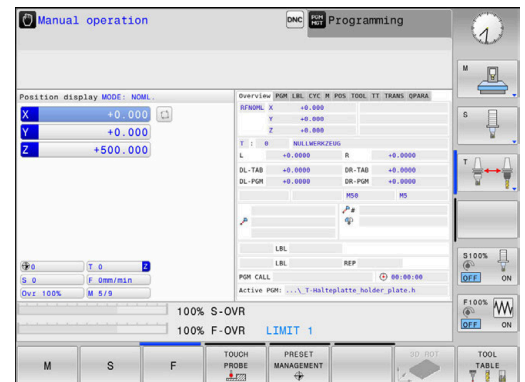
Caution: Danger for the operator!

Machines and machine components always present 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 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.

CE

- ▶ Press the **CE** key
- ▶ The control compiles the PLC program.



- ▶ 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 130
- Operating modes
Further information: "Programming", Page 52

2.3 Graphically testing the workpiece

Selecting the Test Run operating mode

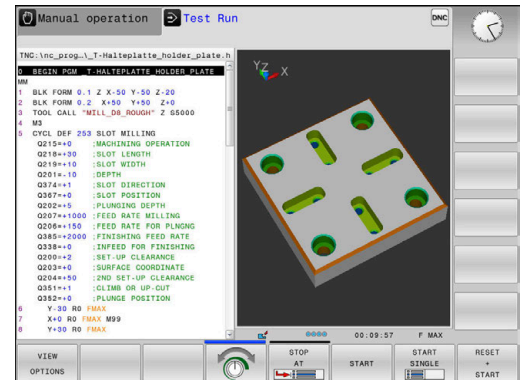
You can test NC programs in the **Test Run** operating mode:



- ▶ 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 52
- Testing NC programs
Further information: "Test run", Page 194



Selecting the tool table

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



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



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



- ▶ 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



- ▶ Move the cursor onto the TOOL.T (active tool table) file, confirm with the **ENT** key: TOOL.T is assigned the status **S** and is therefore active for **Test Run**



- ▶ Press the **END** key: Exit the file manager

Further information on this topic

- Tool management
Further information: "Entering tool data into the table", Page 103
- Testing NC programs
Further information: "Test run", Page 194

Selecting the NC program



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



- ▶ Press the **PROGRAM + MACHINE** soft key
- > In the left half of the screen the control shows the NC program; in the right half it shows the workpiece blank.

The control provides the following views:

Soft keys	Function
	Plan view
	Projection in three planes
	3-D view

Further information on this topic

- Graphic functions
Further information: "Graphics ", Page 182
- Performing a test run
Further information: "Test run", Page 194

Starting the test run



- ▶ 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



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



- ▶ Press the **START** soft key
- > The control resumes the test run after a break

Further information on this topic

- Performing a test run
Further information: "Test run", Page 194
- Graphic functions
Further information: "Graphics ", Page 182
- Adjusting the simulation speed
Further information: "Speed of the setting test runs", Page 187

2.4 Setting up tools

Selecting the Manual operation mode

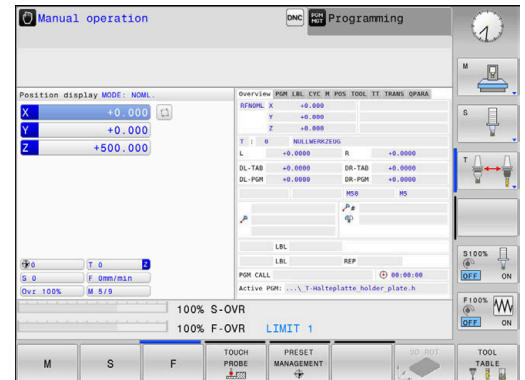
Tools are set up in the **Manual operation** mode:



- ▶ 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 52



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, proceed as follows:



- ▶ Display the tool table
- > The control shows the tool table.
- ▶ Edit the tool table: 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
- ▶ To exit the tool table, press the **END** key

T	NAME	L	R	R2	DL	M
0	NULL WERKZEUG	0	0	0	0	
1	D2	30	1	0		
2	D4	40	2	0		
3	D6	50	3	0		
4	D8	60	4	0		
5	D10	60	5	0		
6	D12	60	6	0		
7	D14	70	7	0		
8	D16	80	8	0		
9	D18	90	9	0		
10	D20	90	10	0		
11	D22	90	11	0		
12	D24	90	12	0		
13	D26	90	13	0		
14	D28	100	14	0		
15	D30	100	15	0		
16	D32	100	16	0		
17	D34	100	17	0		
18	D36	100	18	0		
19	D38	100	19	0		

Further information on this topic

- Operating modes of the control
Further information: "Modes of operation", Page 52
- Working with the tool table
Further information: "Entering tool data into the table", Page 103

Editing the TOOL_PTCH pocket table



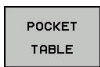
Refer to your machine manual.

The function of the pocket table depends on the machine.

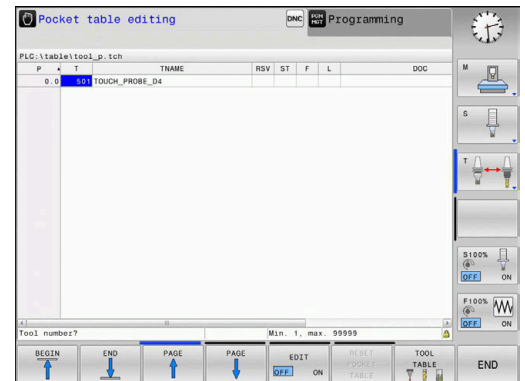
In the pocket table TOOL_PTCH (permanently saved under **TNC:\table**) you specify which tools your tool magazine contains. To enter data in the pocket table TOOL_PTCH, proceed as follows:



- ▶ Display the tool table
- > The control shows the tool table.



- ▶ Display the pocket table
- > The control shows the pocket table.
- ▶ Edit the pocket table: 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
- ▶ To leave the pocket table, press the **END** key



Further information on this topic

- Operating modes of the control
Further information: "Modes of operation", Page 52
- Working with the pocket table
Further information: "Pocket table for tool changer", Page 110

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** operating mode
Further information: "Moving the machine axes", Page 134

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

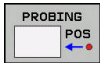
- Presetting with a 3-D touch probe
Further information: "Presetting with a 3-D touch probe (option number 17)", Page 174
- Presetting without 3-D touch probe
Further information: "Presetting without a 3-D touch probe", Page 158

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

- ▶ Insert a 3-D touch probe: In the **Positioning w/ Manual Data Input** mode, run a **TOOL CALL** block containing the tool axis and then return to 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 3-D touch probe (option number 17)", Page 174

2.6 Machining the workpiece

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

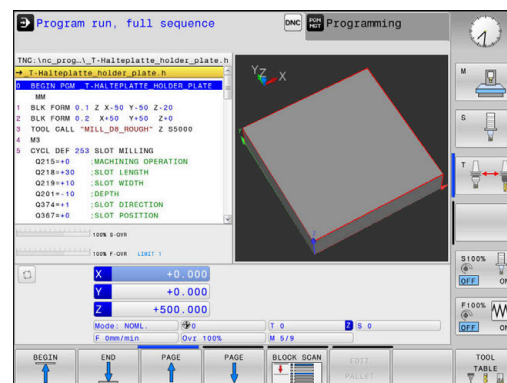
You can run NC programs either in the **Program run, single block** or the **Program run, full sequence** operating mode:



- ▶ 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



- ▶ Press the **Program run, full sequence** key
- ▶ The control switches to the **Program run, full sequence** operating mode and runs the NC program after NC start 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 52
- Running NC programs
Further information: "Program run", Page 199

Selecting the NC program



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



- ▶ 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



- ▶ 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 199

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.

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 Conversational 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 263
- Firewall
Further information: "Firewall", Page 273
- Sandbox
Further information: "Sandbox tab", Page 288
- Integrated browser
Further information: "Displaying Internet files", Page 76
- Management of external access
Further information: "Permitting or restricting external access", Page 245
- Monitoring of TCP and UDP ports
Further information: "Portscan", Page 258
- Remote diagnosis
Further information: "Remote Service", Page 259
- User administration
Further information: "User administration", Page 292

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.

 **DANGER**

Caution: Danger for the operator!

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

HEIDENHAIN found that virus scanners can negatively affect the behavior of the 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.

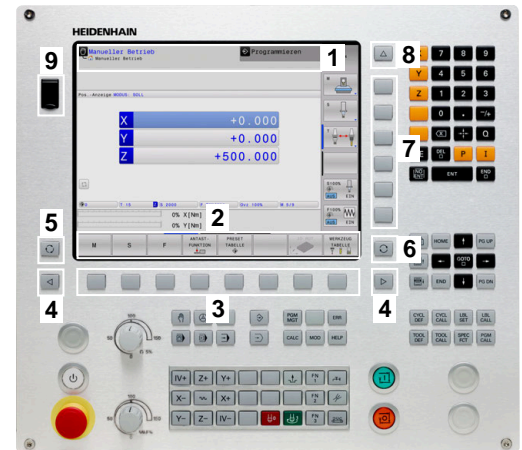
If you insist on using a virus scanner, you must operate the control in a protected network (with a gateway and a virus scanner). The subsequent installation of a virus scanner is not possible.

3.2 Visual display unit and operating panel

Display screen

The control is shipped with a 12.1-inch screen.

- 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 tool builders
- 8 Keys for switching the soft keys for machine tool builders
- 9 USB connection



Setting the screen layout

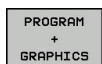
You select the screen layout yourself. In the **Programming** operating mode, for example, you can have the control show the NC program blocks in the left window while the right window displays programming graphics. You could also display the program structure in the right window instead, or display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

Setting the screen layout:



- ▶ Press the **screen layout** key: The soft-key row shows the available layout options

Further information: "Modes of operation", Page 52

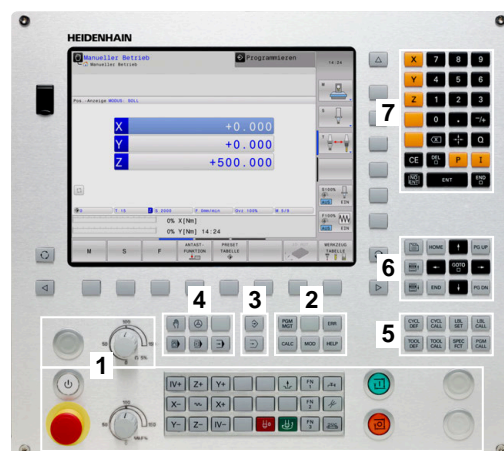


- ▶ Select the desired screen layout with a soft key

Control panel

The TNC 128 is delivered with an integrated operating panel.

- 1 Machine operating panel
More information: Machine manual
- 2
 - File management
 - 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.



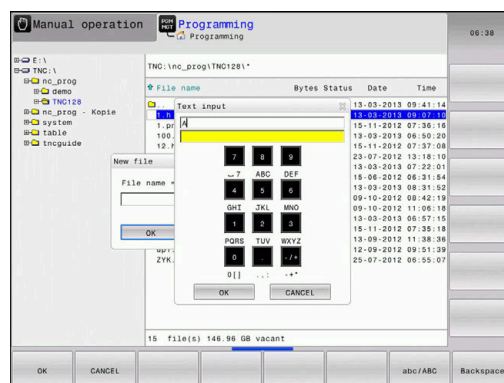
Refer to your machine manual.

Some machine tool builders do not use the standard HEIDENHAIN operating panel.

External keys, e.g. **NC START** or **NC STOP**, are described in your machine manual.




Screen keypad

You can enter letters and special characters with the screen keypad or (if available) with an alphabetic keyboard connected to the USB port.



Entering text with the screen keypad

Proceed as follows to use the screen keypad:

-  ▶ Press the **GOTO** key if you want to enter letters, e.g. a program name or directory name, using the screen keypad.
- ▶ The control opens a window in which the numeric keypad of the control is displayed with the corresponding letters assigned.
-  ▶ Press the numerical key until the cursor is on the desired letter
- ▶ Wait until the control transfers the selected character before you enter the next character
-  ▶ Use the **OK** soft key to load the text into the open dialog field

Use the **abc/ABC** soft key to select upper or lower case. If your machine manufacturer has defined additional special characters, you can call them with the **SPECIAL CHARACTERS** soft key and insert them. Use the **BACKSPACE** soft key to delete individual characters.

3.3 Modes of operation

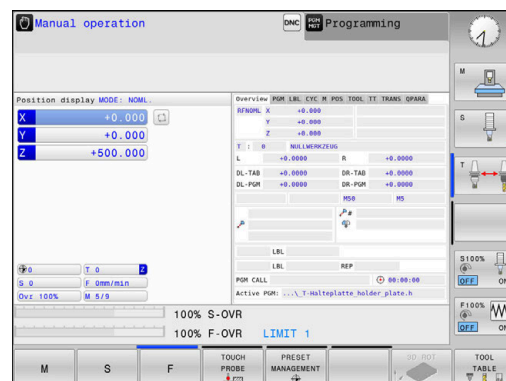
Manual Operation and El. Handwheel

The **Manual operation** mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments and set the presets.

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

Soft keys for the screen layout (select as described above)

Soft key	Window
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display
POSITION + WORKPIECE	Left: positions, right: workpiece

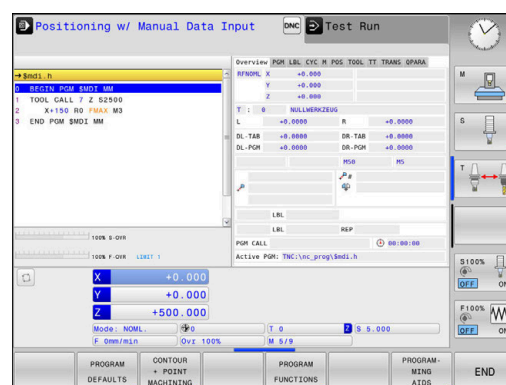


Positioning with Manual Data Input

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

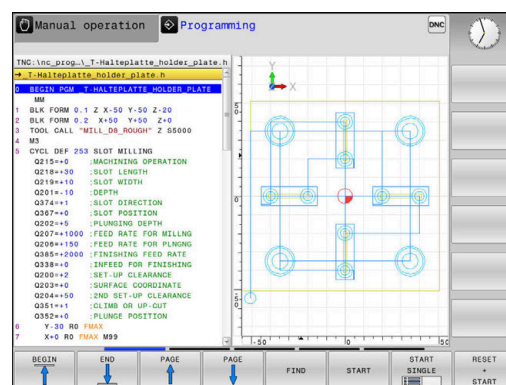


Programming

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

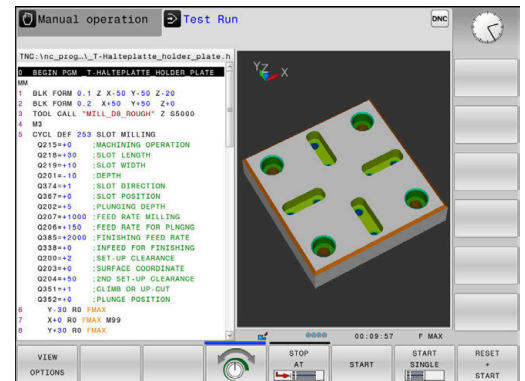


Test Run

In the **Test Run** operating mode, the control checks NC programs and program sections 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



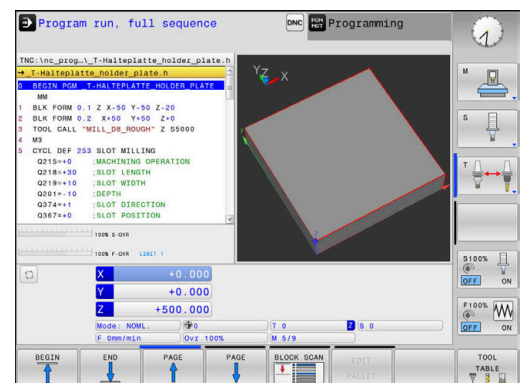
Program Run, Full Sequence and Program Run, Single Block

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.

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.

It is displayed automatically in the following operating modes:

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

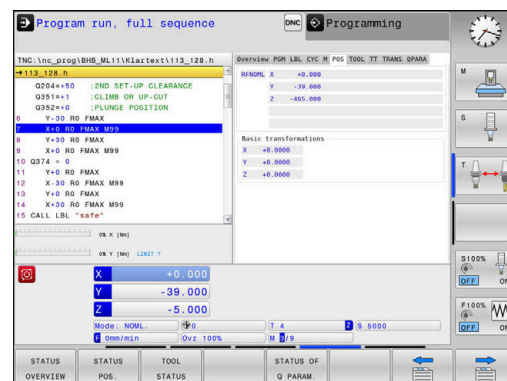






If the **GRAPHICS** screen layout is selected the status display is not shown.


In the **Manual operation** and **Electronic handwheel** operating modes, the status display appears in the large window.

Information in the status display

Icon	Meaning
ACTL	Position display mode, e.g. actual or nominal coordinates of the current position
XYZ	Machine axes; the control displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
	Axis is clamped
	Axis can be moved with the handwheel
	Axes are mirrored and moved
	No NC program selected, NC program reselected, NC program aborted via internal stop, or NC program terminated In this condition the control has no modally effective program information (i.e. the contextual reference), so that all handling is possible, e.g. cursor movements or modification of Q parameters.
	NC program started, execution runs For safety reasons, the control permits no handling in this condition



Icon	Meaning
	<p>NC program stopped, e.g. in Program run, full sequence operating mode after pressing the NC stop key</p> <p>For safety reasons, the control permits no handling in this condition</p>
	<p>NC program interrupted, e.g. in Positioning w/ Manual Data Input operating mode, following the error-free execution of an NC block</p> <p>In this condition the control permits various handling, e.g. cursor movements or the modification of Q parameters. With this handling the control may lose the modally effective program information (i.e. the contextual reference). Loss of this contextual reference may cause undesired tool positions!</p> <p>Further information: "Positioning w/ Manual Data Input operating mode", Page 219 and "Program-controlled interruptions", Page 205</p>
	<p>NC program aborted or terminated</p>
	<p>Pulsing spindle speed function is active</p>





The order of icons can be changed with the optional machine parameter **iconPrioList** (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 the program run. They can be called in all operating modes. Exception: the **Programming operating mode**. In the **Test Run** operating mode, only a limited status display is available.

To switch on the additional status display

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

To select an additional status display



- ▶ Toggle through the soft key rows until the **STATUS** soft keys appear



- ▶ Either select the additional status display directly with the soft key, e.g. positions and coordinates; or



- ▶ use the switch-over soft keys to select the desired view

Select the status displays described below as follows:

- directly with the corresponding soft key
- via the switchover 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.

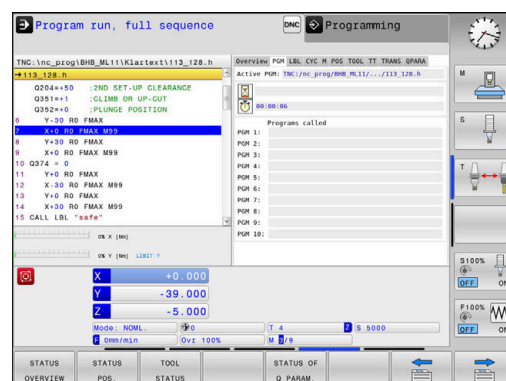
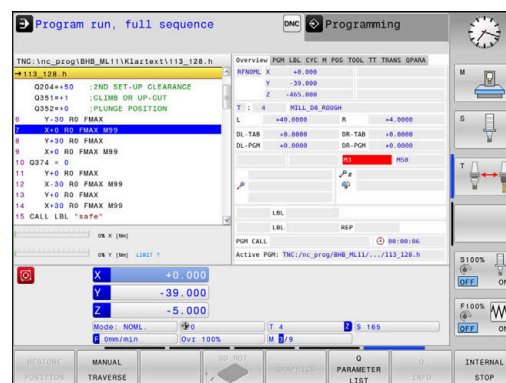
Overview

The **Overview** status form is displayed by the control following switch-on if you selected the screen layout **PROGRAM + STATUS** (or **POSITION + STATUS**). 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
	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	NC program called with PGM CALL
	Current machining time
	Name and path of the active main program

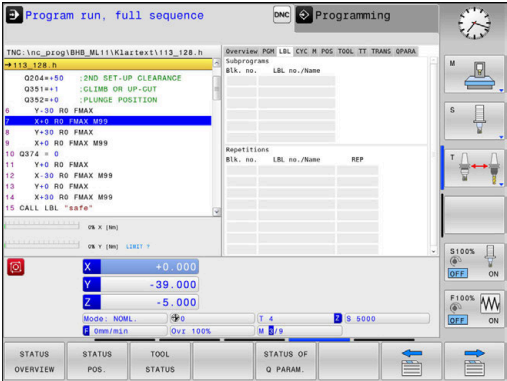
General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name and path of the active main program
	Actual/nominal value counter
	Dwell time counter
	Current machining time
	Called NC programs



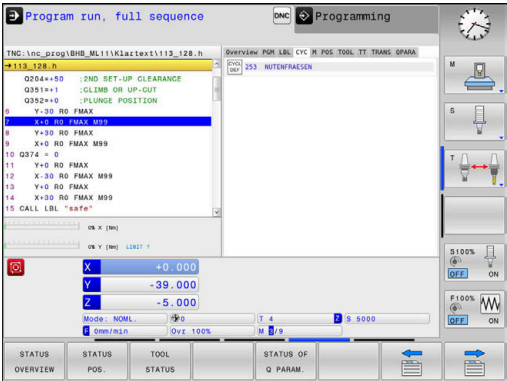
Program section repeats and subprograms (LBL tab)

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 block number in which the subprogram was called and the label number that was called



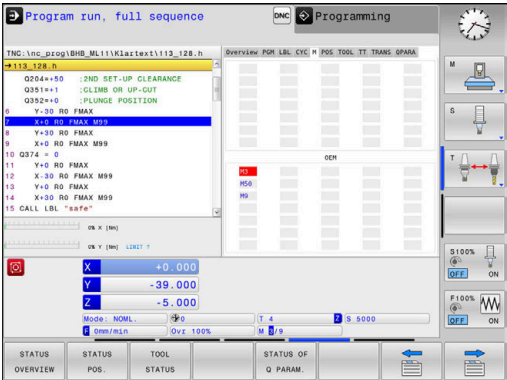
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle



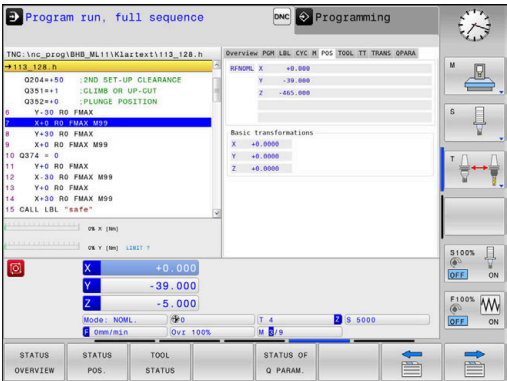
Active miscellaneous functions M (M tab)

Soft key	Meaning
No direct selection possible	List of the active M functions with fixed meaning
	List of the active M functions that are adapted by your machine manufacturer




Positions and coordinates (POS tab)

Soft key	Meaning
<div>STATUS POS.</div>	Type of position display, e.g. actual position



Information on tools (TOOL tab)

Soft key	Meaning
	Display of active tool: <ul style="list-style-type: none"> ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Oversizes (delta values) from the tool table (TAB) and the 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)



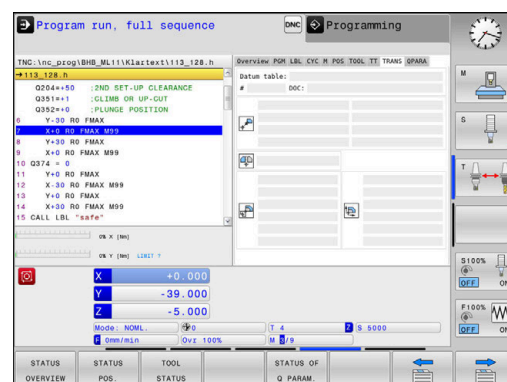
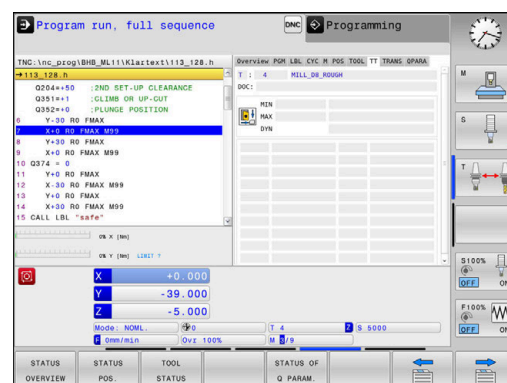
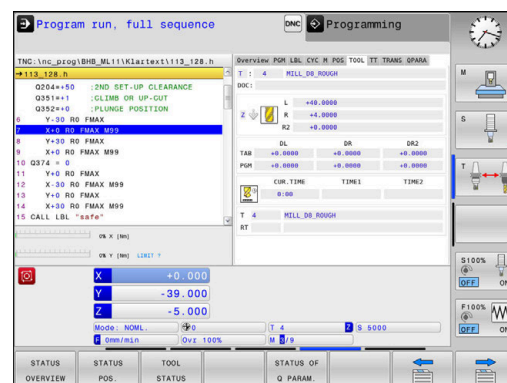
The control displays this tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Active tool
	Measured values from tool measurement

Coordinate transformations (TRANS tab)

Soft key	Meaning
No direct selection possible	Name of the active datum table
	Active datum number (#), comment from the active line of the active datum number (DOC) from Cycle 7
	Active datum shift (Cycle 7); the control displays an active datum shift in up to 3 (5) axes
	Mirrored axes (Cycle 8)
	Active scaling factor/factors (Cycle 11 / 26); the control displays an active scaling factor in up to 6 axes
	Scaling datum

Further information: Conversational Programming User's Manual

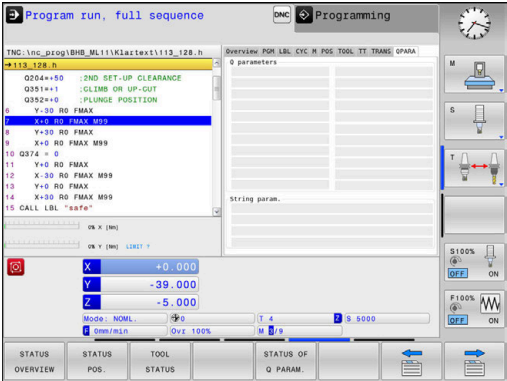


Displaying Q parameters (QPARA tab)

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

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 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, whereby e-08 corresponds to the factor of 10^{-8} .



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**.



Depending on the setting, the control generates backup files with the extension *.bak after editing and saving of 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. Table names must start with a letter.



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 62

Displaying externally generated files on the control

The control features several additional 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
Graphics files	bmp
	gif
	jpg
	png

Further information: "Additional tools for management of external file types", Page 73

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 \.



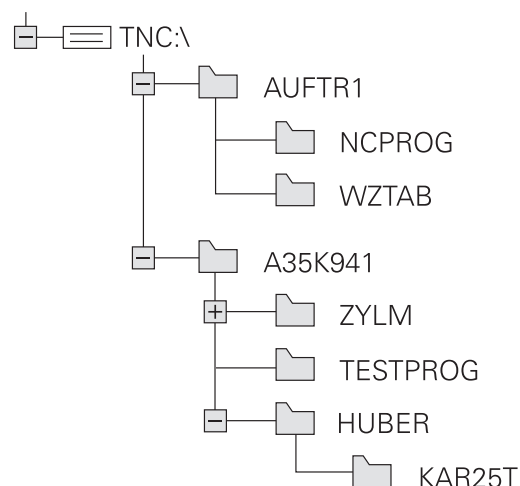
The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

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.



Calling the file manager

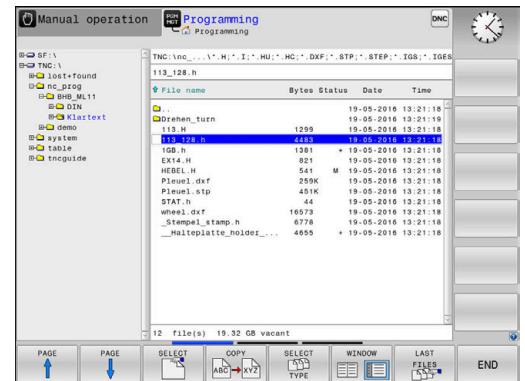


- Press the **PGM MGT** key
- The control displays the file management window (see figure for default setting. If the control displays a different screen layout, press the **WINDOW** soft key).

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	File is selected in the Programming operating mode
S	File is selected in the Test Run operating mode
M	The file is selected in a Program Run operating mode
+	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 erasing and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

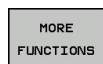


To display the dependent files, set the machine parameter **dependentFiles** (no. 122101) to **MANUAL**.

Additional functions

Protecting a file and canceling file protection

- Place the cursor on the file you want to protect



- Select the additional functions:
Press the **MORE FUNCTIONS** soft key



- Activate file protection:
Press the **PROTECT** soft key



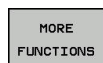
- The file is tagged with the "protected" symbol.



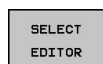
- Cancel file protection:
Press the **UNPROTECT** soft key

Selecting the editor

- Place the cursor on the file you want to open



- Select the additional functions:
Press the **MORE FUNCTIONS** soft key

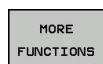


- Select the editor:
Press the **SELECT EDITOR** soft key
- Mark the desired editor
 - **TEXT EDITOR** for text files, e.g. **.A** or **.TXT**
 - **PROGRAM EDITOR** for NC programs **.H** and **.I**
 - **TABLE EDITOR** for tables, e.g. **.TAB** or **.T**
 - **BPM EDITOR** for pallet tables **.P**
- Press 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, proceed as follows:



- 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 68

Selecting drives, directories and files



- To call the file manager, press 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-hand 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 soft key for the desired file type, or



- Display all files: Press the **SHOW ALL** soft key, or



- Use wildcards, e.g. **4*.h**: Show all files of type .h starting with a 4

- 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.



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.

Choosing one of the last files selected



- ▶ To call the file manager, press the **PGM MGT** key.



- ▶ 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:



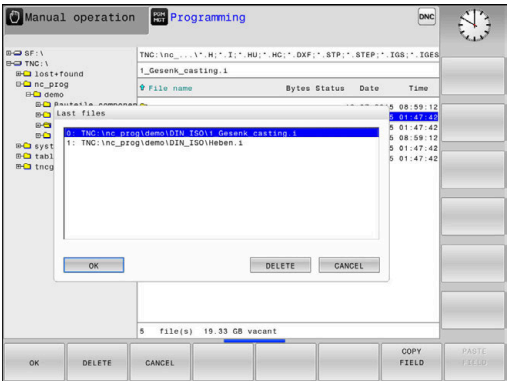
- ▶ Moves the cursor up and down within a window




- ▶ Select the file: Press the **OK** soft key, or



- ▶ Press the **ENT** key





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



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
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The control automatically detects these types of USB devices when connected. The control does not support USB devices with other file systems (such as NTFS). The control displays the **USB: TNC does not support device** error message when such a device is connected.



If an error message is displayed when connecting a USB data medium, check the setting in the SELinux security software.

Further information: "SELinux security software", Page 263

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 correctly detect a USB device with the FAT/VFAT or exFAT file system, 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 tool builder 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 a dialog **Write access on USB device** 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, proceed as follows:



- ▶ Move the cursor to the left-hand window
- ▶ Press the **MORE FUNCTIONS** soft key
- ▶ Remove the USB device



Data transfer to or from an external data carrier



Before you can transfer data to an external data medium, you must set up the data interface.

Further information: "Setting up data interfaces", Page 276



- ▶ Press the **PGM MGT** key



- ▶ 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 from the control to the external data medium, 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.



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



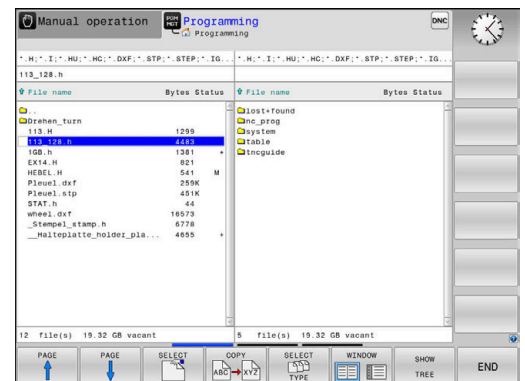
- ▶ Press the **SHOW FILES** soft key
- ▶ Use the arrow keys to select the desired file
- ▶ Press the **COPY** soft key



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



- ▶ 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 in a network



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



Use the Ethernet card to connect the control to the network.

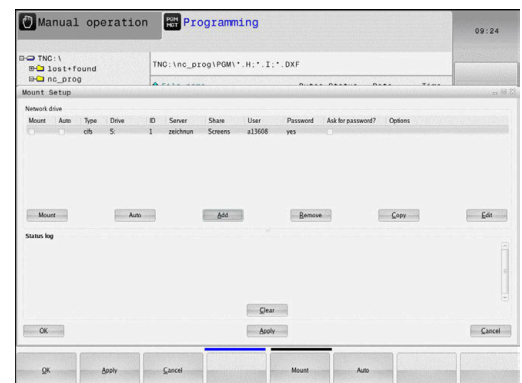
Further information: "Ethernet interface ", Page 282

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

If the control is connected to a network, the left directory window displays additional drives. All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.



The control can also run NC programs directly from a network drive. The external drive does not provide write protection, however. This may lead to problems caused by data transfer or by a change to the NC program during machining.



Connecting and disconnecting a network drive

- ▶ Press the **PGM MGT** key



- ▶ Press the **NET** soft key
- ▶ Press the **DEFINE NETWORK CONNECTN.** soft key.
- ▶ In a window the control shows the network drives available for access.
- ▶ Use the soft keys described below to define the connections for each drive

Soft key	Function
Connect	Establish the network connection. If the connection is active, the control marks the Mount column.
Separate	End network connection
Auto	Automatically establish network connection whenever the control is switched on. The control marks the Auto column if the connection is established automatically
Add	Set up new network connection
Remove	Delete existing network connection
Copy	Copy network connection
Edit	Edit network connection
Clear	Delete the status window

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 270

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.



Take the time occasionally to delete any unneeded files so that the control always has enough hard-disk space for system files (such as the tool table).

Importing a file from an iTNC 530



Refer to your machine manual.

The machine tool builder can adapt the **ADAPT NC PGM / TABLE** function.

The machine tool builder 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 tool builder 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 108

Additional tools for management of external file types

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


File types	Description
PDF files (pdf)	Page 74
Excel spreadsheets (xls, csv)	Page 75
Internet files (htm, html)	Page 76
ZIP archives (zip)	Page 78
Text files (ASCII files, e.g. txt, ini)	Page 79
Video files (ogg, oga, ogv, ogx)	Page 80
Graphic files (bmp, gif, jpg, png)	Page 80



Files with the extensions pdf, xls, zip, bmp, gif, jpg and png must be transmitted in binary format from the PC to the control. Adjust the setting in the **TNCremo** software, if required (menu item >**Extras** > **Configuration** > **Mode**).

Displaying PDF files

To open PDF files directly on the control, proceed as follows:

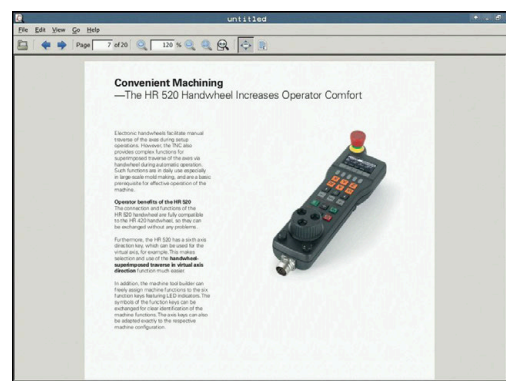
- 
 - ▶ Call the file manager: Press the **PGM MGT** key
 - ▶ Select the directory in which the PDF file is saved
 - ▶ Move the cursor to the PDF file
 - ▶ Press the **ENT** key
 - ▶ The control opens the PDF file in its own application using the **PDF viewer** additional tool.



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.



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 in **Help**.



Proceed as follows to exit the **PDF viewer**:

- ▶ 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, proceed as follows to close the **PDF viewer**:



- ▶ Press the key for switching the soft keys
- ▶ The **PDF viewer** opens the **File** pull-down menu.





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





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

Displaying and editing Excel files

Proceed as follows to open and edit Excel files with the extension **xls**, **xlsx** or **csv** directly on the control:

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




 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.

 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 in **Help**.

Proceed as follows 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, proceed as follows to close the additional **Gnumeric** tool:

- 
 - ▶ Press the key for switching the soft keys
 - ▶ The **Gnumeric** additional tool opens the **File** pull-down menu.
- 
 - ▶ Move the cursor to the **Close** menu item
- 
 - ▶ Press the **ENT** key
 - ▶ The control returns to the file manager.

Displaying Internet files



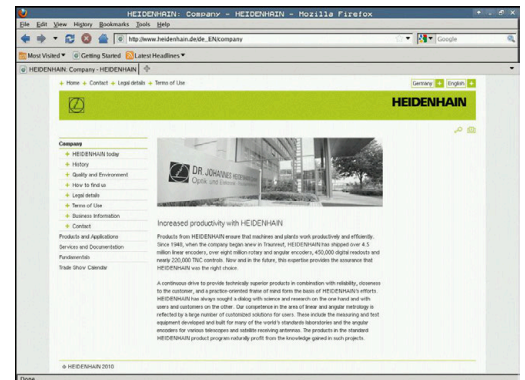
The network must ensure protection against viruses and malicious software. The same applies to access to the Internet or other networks.

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



Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox.

Further information: "Sandbox tab", Page 288



Proceed as follows 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** additional tool.



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.



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 **Web Browser** is available in **Help**.

When you start 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 time and you are connected to the Internet.



Reactivate SELinux after running the update.

Proceed as follows to close 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, proceed as follows to close the **web browser**:



- ▶ Press the key for switching the soft keys: The **web browser** opens the **File** pull-down menu



- ▶ Move the cursor to the **Quit** menu item



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

Working with ZIP archives

Proceed as follows 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
- ▶ Press the **ENT** key
- ▶ The control opens the archive file in its own application using the **Xarchiver** additional tool.

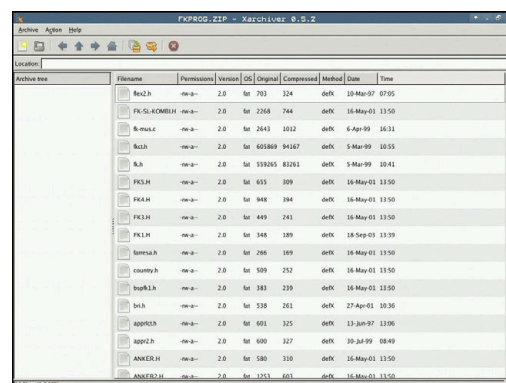
ENT



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 the **Xarchiver** function is provided in **Help**.



To exit **Xarchiver**, proceed as follows:

- ▶ Use the mouse to select the **ARCHIVE** menu item
- ▶ Select **Exit**
- ▶ The control returns to the file manager.

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



- ▶ Press the key for switching the soft keys
- ▶ **Xarchiver** opens the **ARCHIVE** pull-down 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
- ▶ Press the **ENT** key
- ▶ The control opens the text file with the internal text editor.

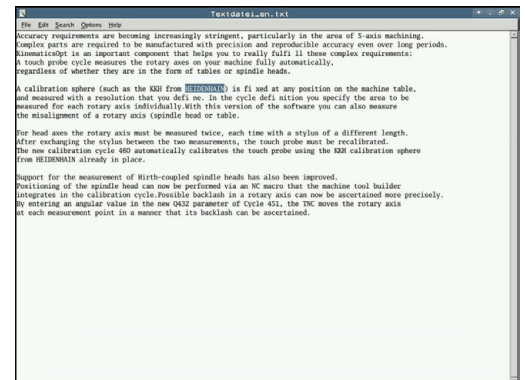
ENT



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



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.



Proceed as follows 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 pull-down menu

Proceed as follows to exit **Leafpad**:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Exit**
- ▶ The control returns to the file manager.

Displaying video files



This function must be enabled and adapted by the machine tool builder.

Proceed as follows 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
- ▶ Press the **ENT** key
- ▶ The control opens the video file in its own application.

ENT



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



Any additional software is installed by your machine tool builder.

Displaying graphics files

Proceed as follows 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
- ▶ Press the **ENT** key
- ▶ The control opens the graphics file in its own application using the **Ristretto** additional tool.

ENT



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** is provided in **Help**.



Proceed as follows to exit **Ristretto**:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select **Exit**
- > The control returns to the file manager.

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



- ▶ Press the key for switching the soft keys
- > **Ristretto** opens the **File** pull-down menu.



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



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

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



The control uses different colors for different error classes:

- red for errors
- yellow for warnings
- green for notes
- blue for information

Long and multi-line error messages are displayed in abbreviated form. Complete information on all pending errors is shown in the error window.

The control displays an error message in the header until it is cleared or replaced by a higher-priority error (higher error class). Information that appears only briefly is always displayed.

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.

If a rare **processor check error** should occur, the control automatically opens the error window. You cannot correct such an error. Shut down the system and restart the control.

Opening the error window

ERR

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

Closing the error window

END

- ▶ Press the **END** soft key, or

ERR

- ▶ Press the **ERR** key
- > The control closes the error window.

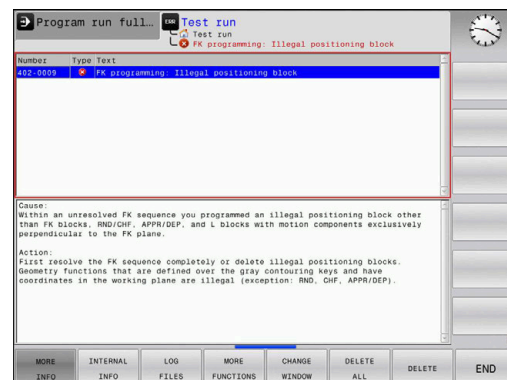
Detailed error messages

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

- Open the error window



- Information on the error cause and corrective action: Position the cursor on the error message and press the **MORE INFO** soft key
- The control opens a window with information on the error cause and corrective action.
- Leave Info: Press the **MORE INFO** soft key again



Soft key: INTERNAL INFO

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

- Open the error window

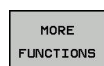


- Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key
- The control opens a window with internal information about the error.
- Exit the details: Press the **INTERNAL INFO** soft key again

Soft key FILTER

The **FILTER** soft key enables you to filter identical warnings listed immediately in succession.

- Open the error window



- Press the **MORE FUNCTIONS** soft key



- Press the **FILTER** soft key. The control filters the identical warnings



- Exit the filter: Press the **GO BACK** soft key

Clearing errors

Clearing errors outside of the error window



- ▶ Clear the errors/messages in the header: Press the **CE** key



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



- ▶ Clear individual error messages: Position the cursor on the error message and press the **DELETE** soft key.



- ▶ Clear all error messages: 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. If the log is full, the control uses a second file. If 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.



- ▶ Press the **LOG FILES** soft key



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



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







- ▶ 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. If the keystroke log is full, the control switches to a second keystroke log. If 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.

- 
 - ▶ Press the **LOG FILES** soft key
- 
 - ▶ Open the keystroke log file: Press the **KEYSTROKE LOG** soft key
- 
 - ▶ Set the previous keystroke log if required: Press the **PREVIOUS FILE** soft key
- 
 - ▶ 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
	Go to beginning of keystroke log
	Go to end of keystroke log
	Find text
	Current keystroke log
	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.

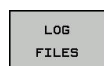
Saving service files

If necessary, you can save the current status 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).

If you repeat the **SAVE SERVICE FILES** function with the same file name, the previously saved group of service files is overwritten. Therefore, use another file name when executing the function another time.

Saving service files

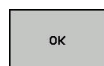
- Open the error window



- Press the **LOG FILES** soft key



- 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.



- Save the service files: Press the **OK** soft key

Calling the TNCguide help system

You can call the control's help system via soft key. Immediately the help system shows you the same error explanation that you receive by pressing the **HELP** soft key.



Refer to your machine manual.

If your machine manufacturer also provides a help system, the control shows an additional **Machine manufacturer (OEM)** soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.

TNCguide context-sensitive help system

Application



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

Further information: "Downloading current help files", Page 92

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



The control tries to start the TNCguide in the language that you have selected as the conversational language. If the required language version is not available, the control automatically opens the English version.

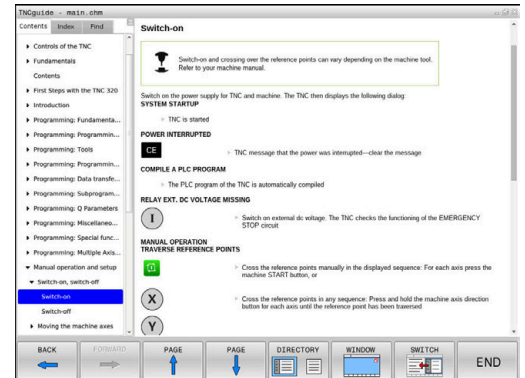
The following user documentation is available in TNCguide:

- Conversational Programming User's Manual (**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, with the contents of all existing .chm files.



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



Working with TNCguide

Calling TNCguide

There are several ways to start the TNCguide:

- ▶ Press the **HELP** key.
- ▶ Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- ▶ 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



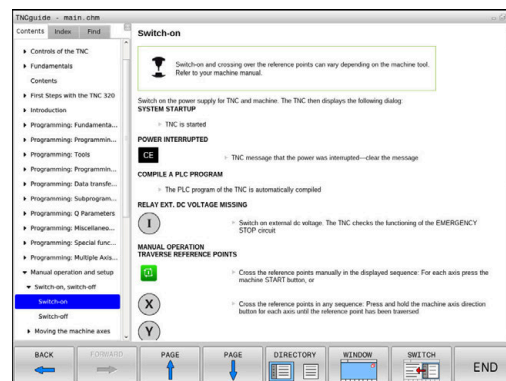
On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

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 the 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 the TNCguide

It's easiest to use the mouse to navigate in TNCguide. A table of contents appears on the left side of the screen. By clicking the rightward pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the 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	Function
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Open up the table of contents ■ If the text window at right is active: No function
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Close the table of contents ■ If the text window at right is active: No function
	<ul style="list-style-type: none"> ■ 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
	<ul style="list-style-type: none"> ■ 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 screen half at right ■ If the text window at right is active: Jump back to the window at left
	<ul style="list-style-type: none"> ■ If the table of contents at left is active: Select the entry above it or below it
	<ul style="list-style-type: none"> ■ If the text window at right is active: Jump to next link
	Select the page last shown
	Page forward if you have used the Select page last shown function
	Move up by one page
	Move down by one page

Soft key	Function
	Display or hide table of contents
	Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window
	The focus is switched internally to the control application so that you can operate the control when the TNCguide is open. If the full screen is active, the control reduces the window size automatically before the change of focus
	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.



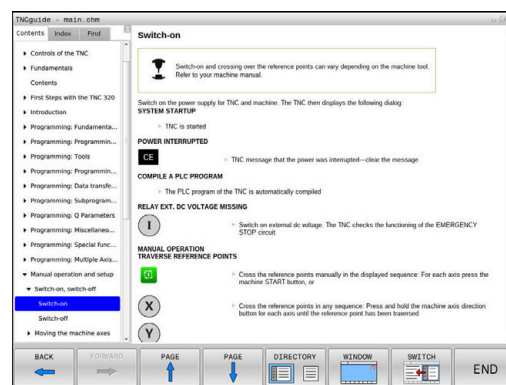
- ▶ Select the **Index** tab
- ▶ Use the arrow keys or the mouse to select the desired keyword

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



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 homepage:

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, e.g. TNC 128 (77184x-07)
- ▶ Select the desired language version from the **TNCguide online help** 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



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
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

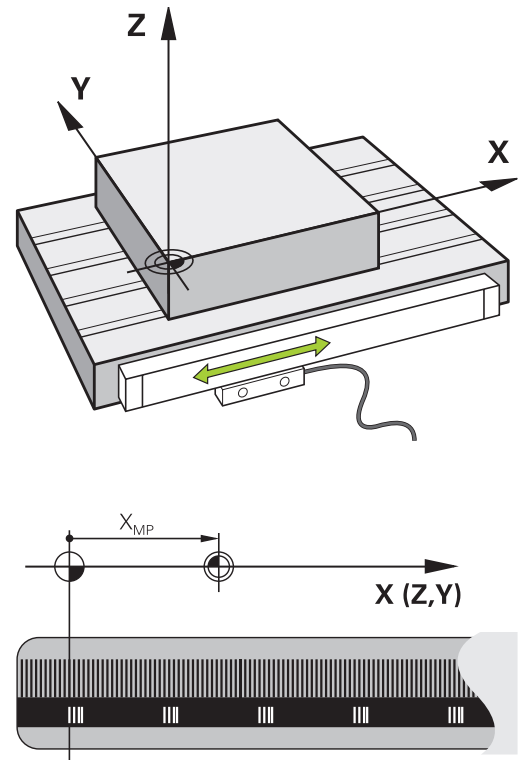
Position encoders and reference marks

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 3-D touch probes and electronic handwheels

3-D touch probes

Applications for HEIDENHAIN 3-D 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 via a cable.

HEIDENHAIN touch trigger probes feature a wear-resistant optical switch that detects the deflection of the stylus. 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 550FS handwheels are available from HEIDENHAIN in addition to the HR 130 and HR 150 panel-mounted handwheels.

Further information: "Moving with the electronic display handwheels", Page 137



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.



Permitted special 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

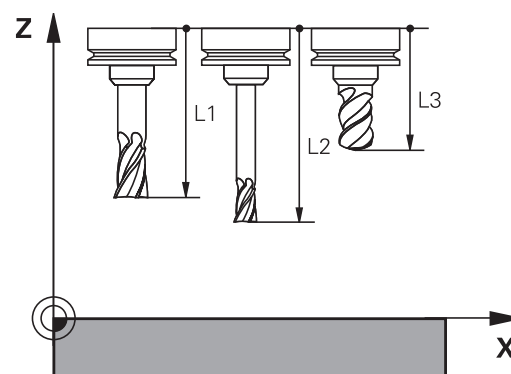
The control automatically replaces lowercase letters with corresponding uppercase letters during saving.

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 $R=0$.

Tool length L

You should always enter the tool length L as an absolute value based on the tool reference point.



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 100

- If your machine has an automatic tool changer
- If you want to work with fixed 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.

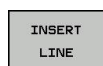
You can select the table view with the **Screen Layout** key. You can choose between a list view and a form view.

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 line of the tool table. Each line 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 lines), add an indexed tool number (such as **T 5.1**) to an existing tool definition (**T 5**). Each additional line 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 line 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.

Proceed as follows to create an indexed tool number (table line):



- ▶ 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 lines to add
- ▶ Enter the original tool number into the **Tool number** input field
- ▶ Confirm with **OK**
- ▶ The control adds the additional lines 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 arrow 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 tool builder adapts the features of the filter function to the requirements of your machine.

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

Use a connected mouse or the navigation keys to work in 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 drop-down menus



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.

Entering tool data into the table

Standard tool data

Abbr.	Inputs	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 – if available – as replacement tool (RT : for R eplacement T ool) An empty field or input 0 means no replacement tool has been defined.	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 (if the tool axis is specified)	Max. tool age for TOOL CALL?
CUR_TIME	Current age of the tool in minutes: The control automatically counts the current tool life (CUR_TIME : For CUR rent TIME) A starting value can be entered for used tools	Current tool age?
TYPE	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?
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]

Abbr.	Inputs	Dialog
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. Used by tapping cycles (Cycle 206, Cycle 207 and Cycle 209). A positive algebraic sign means a right-hand thread.	Tool thread pitch?
LAST_USE	Date and time that the tool was last inserted via TOOL CALL	Date/time of last tool call
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: "Allocating parameterized tool carriers", Page 127	Tool-carrier kinematics
OVRTIME	Time for exceeding the tool life in minutes Further information: "Overtime for tool life", Page 113 Function is defined by the machine manufacturer. Refer to your machine manual.	Tool life expired

Tool data for automatic tool measurement



Refer to your machine manual.

The machine tool builder defines whether the **R-OFFS** offset will be taken into account for a tool with **CUT 0**.

Your machine tool builder defines the default value for the **R-OFFS** and **L-OFFS** columns.

Abbr.	Inputs	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 0.9999 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 0.9999 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	Tool length measurement: Tool offset between stylus center and tool center.	Tool offset: radius?
L-OFFS	Radius measurement: Tool offset between upper surface of stylus and lower surface of tool in addition to offset-ToolAxis .	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 3.2767 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 0.9999 mm	Breakage tolerance: radius?

Editing the tool table

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** modes the control also uses the TOOL.T tool table. In the **Test Run** mode, press the **TOOL TABLE** soft key to edit it.

To open the tool table TOOL.T:

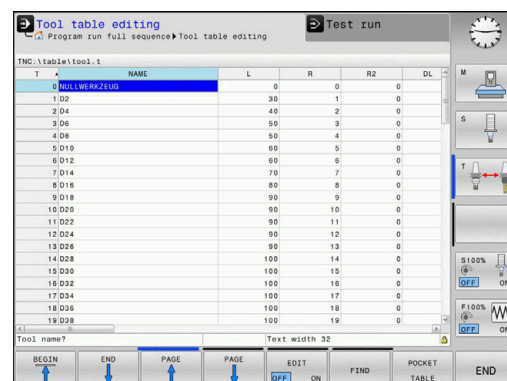
- Select any machine operating mode



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



- Set the **EDIT** soft key to **ON**



















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.

Proceed as follows 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
- Mouse click on a cell: move to the cell
- Double click on a cell: place the cursor in the cell; with selection fields: open the selection dialog

Soft key	Editing functions of the tool table
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Find the text or number
	Go to beginning of line

Soft key	Editing functions of the tool table
	Go to end of line
	Copy active field
	Insert copied field
	Add the entered number of lines (tools) at the end of the table
	Insert a line with definable tool number
	Delete the current line (tool)
	Sort the tools according to the content of a column
	Select possible entries from a pop-up window
	Reset the value
	Place the cursor in the current cell

Importing tool tables



Refer to your machine manual.

The machine tool builder can adapt the **ADAPT NC PGM / TABLE** function.

The machine tool builder 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



- ▶ Press the **PGM MGT** key



- ▶ Move the cursor to the tool table you want to import



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



- ▶ 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, a comma is converted to a period.

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 transferred as well. Tool types not present are imported as type **Undefined**. Check the tool table after the import.

Overwriting tool data from an external PC

Application

The **TNCremo** software provides an especially convenient way to use an external PC to overwrite tool data.

Further information: "Software for data transfer", Page 280

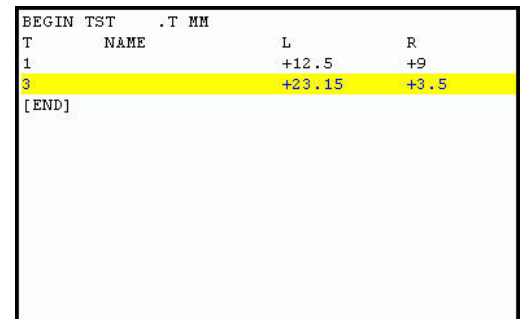
This application case occurs if you wish to determine tool data on an external tool presetter and then transfer this to the control.

Requirements

In addition to option 18 HEIDENHAIN DNC, **TNCremo** (version 3.1 or higher) is required with **TNCremoPlus** functions.

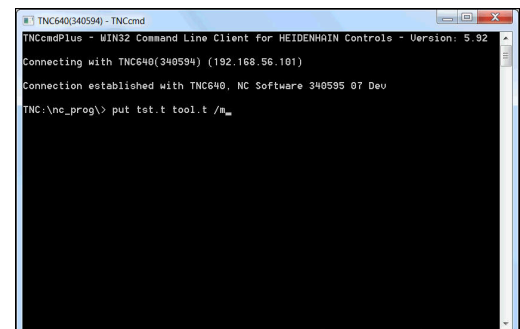
Procedure

- ▶ Copy the TOOL.T tool table to the control, for example to TST.T
- ▶ Start the **TNCremo** data transfer software on the PC
- ▶ Connect to the control
- ▶ Transfer the copied tool table TST.T to the PC
- ▶ Use any text editor to reduce TST.T to the lines and columns to be changed (see figure). Make sure that the header is not changed and the data is always flush in the column. The tool number (column T) need not be consecutive
- ▶ In **TNCremo**, select the menu item <Extras> and <TNCcmd>: This starts TNCcmd
- ▶ To transfer the TST.T file to the control, enter the following command and confirm with the return key (see figure):
put tst.t tool.t /m



T	NAME	L	R
1		+12.5	+9
3		+23.15	+3.5

[END]



```

TNC640(340594) - TNCcmd
TNCcmdPlus - WIN32 Command Line Client for HEIDENHAIN Controls - Version: 5.92
Connecting with TNC640(340594) (192.168.56.101)
Connection established with TNC640, NC Software 340595 07 Dev
TNC:\nc_prog\> put tst.t tool.t /m
  
```



During transfer, only the tool data defined in the subfile (e.g. TST.T) is overwritten. All other tool data of the table TOOL.T remains unchanged.

The procedure for copying tool tables using the file manager is described in the file management.

Further information: Conversational Programming User's Manual

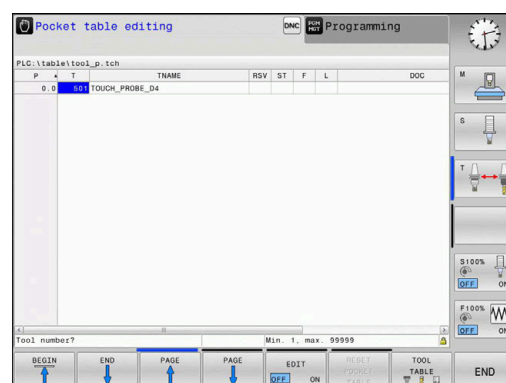
Pocket table for tool changer



Refer to your machine manual.

The machine tool builder adapts the features of the pocket table to the requirements of your machine.

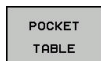
You need a pocket table for automatic tool changing. You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:\table** directory. The machine manufacturer can amend the name, path and content of the pocket table. If you wish, you can select different views 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

Proceed as follows to select the pocket table in the Programming operating mode:



- ▶ To 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

Abbr.	Inputs	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 = NO ENT
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 tool builder. The machine tool documentation provides further information	Value?
PTYP	Tool type. Function is defined by the machine tool builder. 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
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Reset pocket table Depends on optional machine parameter enableReset (no.106102)
	Reset tool number T column Depends on optional machine parameter showResetColumnT (no. 125303)
	Go to beginning of line
	Go to end of line
	Simulate a tool change
	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 the value
	Place the cursor in the current cell
	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 tool builder.

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 tool builder.

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 244

Generating a tool usage file

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

- Completely simulate the 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	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ TOKEN = TOOL: Path name of the active main program or subprogram ■ TOKEN = STOTAL: Path name of the subprogram
T	Tool number with tool index
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	<ul style="list-style-type: none"> ■ 0: The tool number is programmed ■ 1: The tool name is programmed

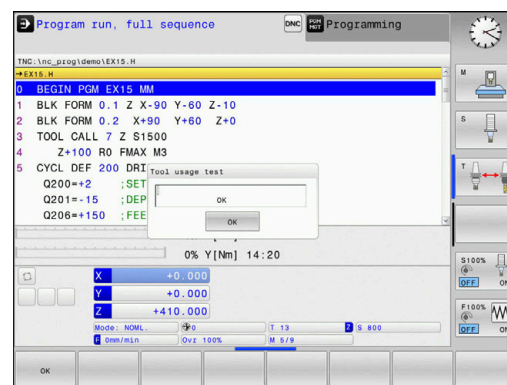
The control saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. This file is not visible unless the machine parameter **dependentFiles** (no. 122101) is set to **MANUAL**.

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 in the tool table with the nominal values from the tool usage file.

- | | |
|--|--|
| <div style="border: 1px solid black; padding: 2px; margin-bottom: 10px; text-align: center;">TOOL
USAGE</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 10px; text-align: center;">TOOL
USAGE
TEST</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 10px; text-align: center;">OK</div> <div style="border: 1px solid black; padding: 2px; text-align: center;">ENT</div> | <ul style="list-style-type: none"> ▶ Press the TOOL USAGE soft key
 ▶ Press the TOOL USAGE TEST soft key > The control opens the Tool usage test pop-up window indicating the result of the usage test.
 ▶ Press the OK soft key > The control closes the pop-up window.
 ▶ Alternative: Press the ENT key |
|--|--|

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



4.2 Tool management

Basics



Refer to your machine manual.

Tool management is a machine-dependent function, which may be partly or completely deactivated. The machine tool builder 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



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**.

The screenshot shows the 'Expanded tool management' window with a table of tool data. The table has columns: T, TYP, NAME, PTYP, TL, POCKET, MAGAZIN, Tool life, and REMARK. The data is as follows:

T	TYP	NAME	PTYP	TL	POCKET	MAGAZIN	Tool life	REMARK
0		Werkzeug	0				Not monitored	
1		MILL_D2_ROUGH	0				Not monitored	
2		MILL_D4_ROUGH	0				Not monitored	
3		MILL_D8_ROUGH	0				Not monitored	
4		MILL_D8_ROUGH	0				Not monitored	
5		MILL_D16_ROUGH	0				Not monitored	
6		MILL_D12_ROUGH	0				Not monitored	
7		MILL_D14_ROUGH	0				Not monitored	
8		MILL_D16_ROUGH	0				Not monitored	
9		MILL_D18_ROUGH	0				Not monitored	
10		MILL_D20_ROUGH	0				Not monitored	
11		MILL_D22_ROUGH	0				Not monitored	
12		MILL_D24_ROUGH	0				Not monitored	
13		MILL_D26_ROUGH	0				Not monitored	
14		MILL_D28_ROUGH	0				Not monitored	
15		MILL_D30_ROUGH	0				Not monitored	
16		MILL_D32_ROUGH	0				Not monitored	
17		MILL_D34_ROUGH	0				Not monitored	
18		MILL_D36_ROUGH	0				Not monitored	
19		MILL_D38_ROUGH	0				Not monitored	

At the bottom of the window, there are buttons for 'BEGIN', 'END', 'PAGE', 'PAGE', 'FORM FOR TOOL', and 'END'. On the right side, there are status indicators for 'S1000' and 'F1000' with 'ON' and 'OFF' buttons.

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.

Expanded tool management

T	Typ	NAME	PTYP	TL	POCKE	MAGAZIN	Tool life	REMAI
1	MILL	MILL_D22_ROUGH	0				Not monitored	
2	MILL	MILL_D4_ROUGH	0				Not monitored	
3	MILL	MILL_D6_ROUGH	0				Not monitored	
4	MILL	MILL_D8_ROUGH	0				Not monitored	
5	MILL	MILL_D10_ROUGH	0				Not monitored	
6	MILL	MILL_D12_ROUGH	0				Not monitored	
7	MILL	MILL_D14_ROUGH	0				Not monitored	
8	MILL	MILL_D16_ROUGH	0				Not monitored	
9	MILL	MILL_D18_ROUGH	0				Not monitored	
10	MILL	MILL_D20_ROUGH	0				Not monitored	
11	MILL	MILL_D22_ROUGH	0				Not monitored	
12	MILL	MILL_D24_ROUGH	0				Not monitored	
13	MILL	MILL_D26_ROUGH	0				Not monitored	
14	MILL	MILL_D28_ROUGH	0				Not monitored	
15	MILL	MILL_D30_ROUGH	0				Not monitored	
16	MILL	MILL_D32_ROUGH	0				Not monitored	
17	MILL	MILL_D34_ROUGH	0				Not monitored	
18	MILL	MILL_D36_ROUGH	0				Not monitored	
19	MILL	MILL_D38_ROUGH	0				Not monitored	

BEGIN END PAGE PAGE FORM FOR TOOL END
















Tool management view

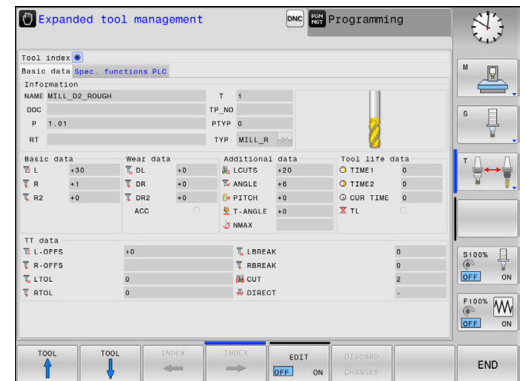
In the new view, the control presents all tool information in the following four tabs:

- **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
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Call the form view of the marked tool. Alternative function: Press the ENT key
	Changing tab: Tools and pockets
	Search function: Here you can select the column to be searched and then the search term either from a list or by entering it
	Import tools
	Export tools
	Delete marked tools
	Add several lines at end of table
	Update table view
	Show the programmed tools column (if the Pockets tab is active)
	Define the settings: <ul style="list-style-type: none"> ■ 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 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
	Select the tool data of the previous tool
	Select the tool data of the next tool
	Select previous tool index (only active if indexing is enabled)
	Select the next tool index (only active if indexing is enabled)
	Open a pop-up window for selection (only available for selection fields)
	Discard all changes made since the form was called
	Add tool index
	Delete tool index
	Copy the tool data of the selected tool
	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 **START** 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



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

4.3 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.

On machines with 3 axes, tool carriers for right-angled angle heads help processing on tool axes **X** and **Y**, as the control takes the dimensions of the angle heads into consideration.

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

- Save tool carrier templates
- Assign input parameters to tool carriers
- Allocate parameterized tool carriers

Saving tool carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. So that you don't have to design all your tool carriers yourself, HEIDENHAIN supplies a range of ready-made tool carrier templates. Tool carrier templates are 3-D models with fixed geometries but changeable dimensions.

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



If the tool carrier templates are not available in your control, please download the data you require from:
<http://www.klartext-portal.com/nc-solutions/en>



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



The tool carrier templates 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!

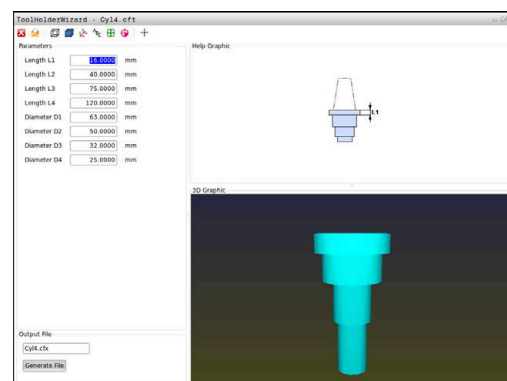
Assigning input parameters to tool carriers

Before the control can factor the tool carrier into the calculations, you must give the tool carrier template 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 a mouse. Using the mouse, you can also set the desired screen layout by drawing a line between the areas **Parameter**, **Help graphics** and **3-D graphics** by holding down the left mouse button.

The following icons are available in the additional **ToolHolderWizard** tool:



Icon	Function
	Close tool
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Return to starting view of the 3-D model



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

Parameterize the tool carrier template in the Manual operation mode

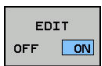
Proceed as follows to parameterize tool carrier templates and save these parameters:



- ▶ Press the **Manual operation** key



- ▶ Press the **TOOL TABLE** soft key



- ▶ Press the **EDIT** soft key



- ▶ Move the cursor to the **KINEMATIC** column



- ▶ Press the **SELECT** soft key



- ▶ Press the **TOOL HOLDER WIZARD** soft key
- > The control opens the additional **ToolHolderWizard** tool in a pop-up window.



- ▶ 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
- ▶ Press the **CLOSE** icon
- > The control closes the additional tool



Parameterize the tool carrier template in the 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



Allocating parameterized tool carriers

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



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

Only use fully parameterized tool carriers!

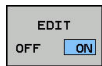
Proceed as follows to allocate a parameterized tool carrier to a tool:



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



- ▶ Press the **TOOL TABLE** soft key



- ▶ Press the **EDIT** soft key



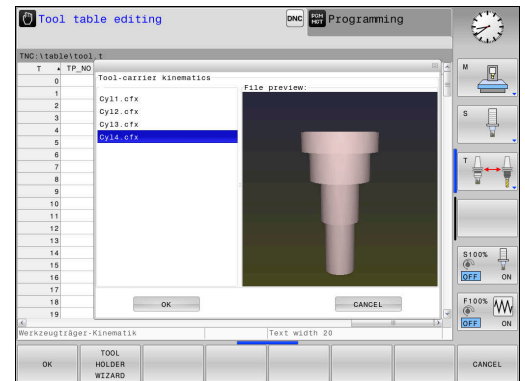
- ▶ Move the cursor to the **KINEMATIC** column of the required tool



- ▶ Press the **SELECT** soft key
- ▶ The control opens a pop-up window with parameterized tool carriers
- ▶ Select the desired tool carrier using the preview screen
- ▶ Press the **OK** soft key
- ▶ The control copies the name of the selected tool carrier to the **KINEMATIC** column



- ▶ Exit the tool table



5

Setup

5.1 Switch-on, switch-off

Switch-on

DANGER

Caution: Danger for the operator!

Machines and machine components always present 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 positions



This section applies only to machine axes with EnDat encoders.

If the actual axis position after the machine is switched on does not match the position at switch-off, 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!

If they are not paid attention to, deviations between the actual axis positions and those expected by the control (saved at the time of switch-off) can lead to undesirable and unforeseeable movements of the axes. There is risk of collision during referencing 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 tool builder

Traversing 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.



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 reference the axes.

You can neither set a preset nor modify a preset via the preset table without having referenced the axes. The control issues the **Traverse reference points** message.

You can traverse the reference points later. For this purpose, in **Manual operation** mode press the **PASS OVER REFERENCE** soft key.

Cross the reference points manually in the displayed sequence:



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

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



- ▶ Press and hold the axis direction button for each axis until the reference point has been traversed



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

Switch-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 not matter what state the control was in!

- ▶ 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



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



- ▶ To move the axis continuously: Press and hold the axis direction button 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 148

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

Incremental jog positioning

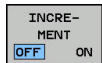
With incremental jog positioning you can move a machine axis by a preset distance.



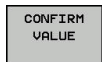
- ▶ Operating mode: Press the **Manual operation** or **Electronic handwheel** key



- ▶ Shift the soft-key row



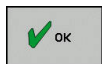
- ▶ Select incremental jog positioning: Switch the **INCREMENT** soft key to **ON**



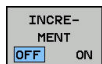
- ▶ Enter the infeed of the **linear axes** and confirm with the **CONFIRM VALUE** soft key



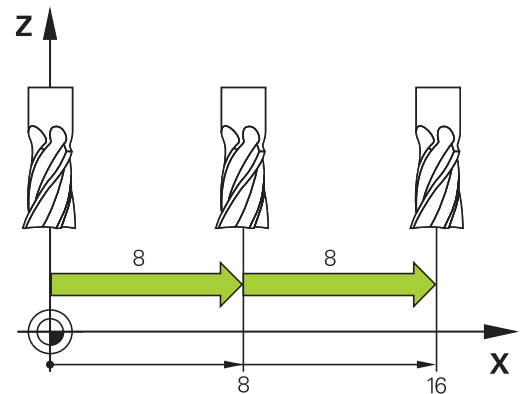
- ▶ Alternatively, confirm with the **ENT** key



- ▶ Confirm with the **OK** soft key
- > The increment is active.



- ▶ Deactivate incremental jog positioning: Switch the **INCREMENT** soft key to **OFF**



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

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

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 tool builder)
- 6 Direction in which the control moves the selected axis
- 7 Machine functions (set by the machine manufacturer)

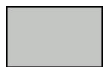


Traversing axes

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

Moving with the electronic display handwheels

⚠ DANGER

Caution: Danger for the operator!

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 new electronic handwheels:

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



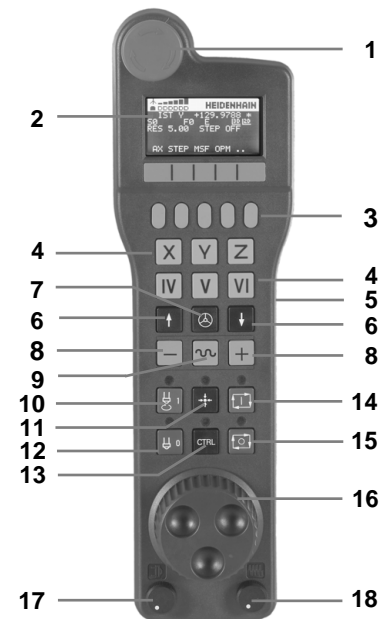
Your machine tool builder can make additional functions of the HR 5xx handwheels available.

The portable HR 520 and HR 550FS handwheels feature a display on which the control shows information. In addition, you can use the handwheel soft keys for important setup functions, e.g. presetting 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 available with the HR 550FS wireless handwheel



Handwheel display

- 1 Only with wireless handwheel HR 550FS:** Shows whether the handwheel is in the docking station or whether wireless operation is active
- 2 Only with wireless handwheel HR 550FS:** Shows the signal strength, 6 bars = maximum signal strength
- 3 Only with wireless handwheel HR 550FS:** Shows the charge status of the rechargeable battery, 6 bars = fully charged 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 *:** STIB (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 message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.
- 10 RES 5.0:** Active handwheel resolution. Path traversed by the selected axis with a 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 wireless handwheel HR 550FS

⚠ DANGER

Caution: Danger for the operator!

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.

If you do not use the HR 550FS, always place it into the handwheel holder intended for this. This way you can ensure that the handwheel batteries are always ready for use thanks to the contact strip on the rear side of the radio handwheel and the recharge control, and that there is a direct contact connection for the emergency stop circuit.

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.

You can operate the HR 550FS with the battery for up to 8 hours before it must be recharged again. If not in use, it is recommended to put the handwheel 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.



When the handwheel is completely discharged, it takes about 3 hours until it is fully recharged in the handwheel holder.

Clean the contacts **1** in the handwheel holder and of the handwheel regularly to ensure their proper functioning.

The transmission range is amply dimensioned. If you should nevertheless happen to come near the edge of the transmission area, which is possible with very large machines, the HR 550FS warns you in time with a plainly noticeable vibration alarm. 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:

► Press the **MOD** key to select the MOD function

► Select **Machine settings**



- Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- Click the **Start handwheel** button to reactivate the radio handwheel
- To save the configuration and exit the configuration menu, press **END**

The **MOD** operating mode includes a function for commissioning and configuring the handwheel.

Further information: "Configuring the HR 550 FS radio handwheel", Page 248

Selecting the axis to be moved

You can activate the principal axes X, Y, Z and three other axes defined by the machine manufacturer directly through the axis keys. Your machine tool builder 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 determines which path an axis takes per revolution of the handwheel. The sensitivity levels are pre-defined and are selectable with the handwheel arrow keys (only when incremental jog is not active).

Selectable sensitivity levels:

0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1 [mm/revolution or degrees/revolution]

Selectable sensitivity levels:

0.00005/0.001/0.002/0.004/0.01/0.02/0.03 [in mm/revolution or degrees/revolution]

Moving the axes

- ▶ To activate the handwheel, press the handwheel button on the HR 5xx
- ▶ Now you can 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 if necessary



- ▶ If required, press and hold the permissive button



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



- ▶ Move the active axis in the positive direction; or



- ▶ Move the active axis in the negative direction



- ▶ To deactivate the handwheel, press the handwheel key on the HR 5xx
- ▶ Now you can operate the control via the operating panel again.

Potentiometer settings

DANGER

Caution: Danger for the operator!

Activating the handwheel does not automatically activate the potentiometers of the handwheel; rather the potentiometers on the operating panel of the control remain active. After an NC start on the handwheel, the control immediately begins with machining or with axis positioning, even though the potentiometers on the handwheel are set to 0 %. There is a risk of death to anybody inside the working space!

- ▶ Before using the handwheel, set the potentiometers of the operating panel to 0 %
- ▶ When using the handwheel, always also activate the potentiometers of the handwheel

The potentiometers of the machine operating panel continue to be active after you have activated the handwheel. If you want to use the potentiometers on the handwheel, proceed as follows:

- ▶ Press the **CTRL** and **handwheel** keys on the HR 5xx at the same time
- > The control shows the soft-key menu for selecting the potentiometers in the handwheel's display.
- ▶ Press the **HW** soft key to activate the handwheel potentiometers

If you have activated the potentiometers on the handwheel, you must reactivate the potentiometers of the machine operating panel before deselecting the handwheel. Proceed as follows:

- ▶ Press the **CTRL** and **handwheel** keys on the HR 5xx at the same time
- > The control shows the soft-key menu for selecting the potentiometers in the handwheel's display.
- ▶ Press the **KBD** soft key to activate the potentiometers of the machine operating panel

The control issues a warning if the handwheel potentiometers are still active after the handwheel has been deactivated.

Incremental jog positioning

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

- ▶ Press the **F2 (STEP)** handwheel soft key
- ▶ To activate incremental jog positioning: Press 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 with soft key 4 (**OK**)
- ▶ With the **+** or **-** handwheel key, move the active handwheel axis in the corresponding direction



If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Inputting miscellaneous functions M

- ▶ Press handwheel soft key **F3 (MSF)**
- ▶ Press handwheel soft key **F1 (M)**
- ▶ Select the desired M function number by pressing the **F1** or **F2** key
- ▶ Execute the M miscellaneous function with the **NC Start** key

Inputting spindle speed S

- ▶ Press handwheel soft key **F3 (MSF)**
- ▶ Press handwheel soft key **F2 (S)**
- ▶ Select the desired speed by pressing the **F1** or **F2** key
- ▶ Activate the new speed S with the **NC Start** key



If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also 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 handwheel soft key **F3 (MSF)**
- ▶ Press handwheel soft key **F3 (F)**
- ▶ Select the desired feed rate by pressing the **F1** or **F2** key
- ▶ Load the new feed rate F with the **F3 (OK)** handwheel soft key



If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Setting a preset

- ▶ Press handwheel soft key **F3 (MSF)**
- ▶ Press the handwheel soft key **F4 (PRS)**
- ▶ If required, select the axis in which the preset is to be set
- ▶ Zero the axis with the handwheel soft key **F3 (OK)** or with **F1** and **F2** set the desired value and then confirm with **F3 (OK)**.
By also pressing the **CTRL** key, you can increase the counting increment to 10

Changing operating modes

With the **F4 (OPM)** handwheel soft key you can use the handwheel to switch the mode of operation if that the current status of the control allows a mode change.

- ▶ 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 tool builder can assign any function to the **Generate NC block** handwheel key.

- ▶ Select **Positioning w/ Manual Data Input** operating mode
- ▶ If required, use the arrow keys on the alphabetic keyboard to select the NC block after which the new positioning block is to 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 the Program Run modes of operation:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during a program interruption (handwheel soft keys **MOP** and then **REPO**). Operate using handwheel soft keys just as you would with the screen soft keys.

Further information: "Returning to the contour", Page 217

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 miscellaneous functions M with soft keys.

Further information: "Enter a miscellaneous function M",
Page 223



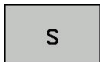
Refer to your machine manual.

The machine tool builder defines which miscellaneous functions are available on the machine and which are allowed in the **Manual operation** mode.

Entering values

Spindle speed S, miscellaneous function M

Proceed as follows to enter the spindle speed:



- ▶ 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

The spindle speed with the entered speed **S** is started with a miscellaneous function **M**. Input a miscellaneous function **M** 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

Proceed as follows to enter the feed rate:



- ▶ Press the **F** soft key
- ▶ The control opens a pop-up window.
- ▶ Enter a feed rate



- ▶ 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 tool builder has defined as minimum feed rate is effective
- If the feed rate entered exceeds the maximum value that has been defined by the machine tool builder, then the value defined by the machine tool builder is effective
- F is not lost during a power interruption
- The control displays the feed rate.

The control shows the current feed rate in the status display.

- If the feed rate is less than 10, the control also shows one of the decimal place that have been 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 lowers the programmed feed rate, 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 speed 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.

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, proceed as follows:



- ▶ Operating mode: Press the **Positioning w/ Manual Data Input** key




- ▶ Press the **F MAX** soft key



- ▶ Enter the desired maximum feed rate
- ▶ Press the OK soft key

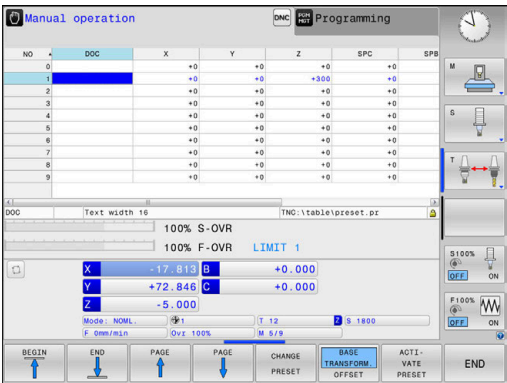
5.4 Preset management

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.

Saving presets in the table



Refer to your machine manual.

The machine tool builder can disable presetting in individual axes.


The machine tool builder can define a different path for the preset table.

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** operating modes only after you have pressed the **CHANGE PRESET** soft key. 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



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



- ▶ Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly



- ▶ Press the **PRESET MANAGEMENT** soft key
- ▶ The control opens the preset table and positions the cursor in the line of the active preset.



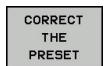
- ▶ Press the **CHANGE PRESET** soft key
- ▶ The control displays all available input options in the soft-key row.



- ▶ 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















- ▶ 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
	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
	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
	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
	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
	Select the table start
	Select the table end
	Select the previous page in the table
	Select the next page in the table
	Select the functions for entry of presets
	Activate the preset of the selected line of the preset table
	Add several lines at end of table
	Copy the field that is currently highlighted
	Insert the copied field
	Reset the selected line: The control enters - into all columns.
	Insert a line as last line in the table
	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

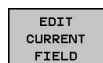
Proceed as follows to protect a preset from being overwritten:



- ▶ Press the **CHANGE PRESET** soft key

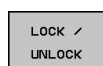


- ▶ Select the **LOCKED** column



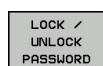
- ▶ Press the **EDIT CURRENT FIELD** soft key

Protection for a preset without using a password:

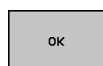


- ▶ Press the **LOCK / UNLOCK** soft key
- ▶ The control writes an **L** in the **LOCKED** column.

Use a password to protect a preset:



- ▶ Press the **LOCK / UNLOCK PASSWORD** soft key



- ▶ Enter the password in the pop-up window
- ▶ 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, proceed as follows:



- ▶ Press the **CHANGE PRESET** soft key

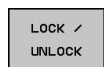


- ▶ Select the **LOCKED** column



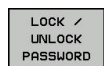
- ▶ Press the **EDIT CURRENT FIELD** soft key

Preset protected without a password:



- ▶ Press the **LOCK / UNLOCK** soft key
- > The control cancels the write-protection.

Preset protected with a password:



- ▶ Press the **LOCK / UNLOCK PASSWORD** soft key



- ▶ Enter the password in the pop-up window
- ▶ Confirm with the **OK** soft key or with the **ENT** key
- > The control cancels the write-protection.

Activating a preset

Activate the 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.

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



Operating notes:

- When activating a preset from the preset table, the control resets any active datum shift, mirroring, or scaling factor.



- ▶ Select the **Manual operation** mode



- ▶ Press the **PRESET MANAGEMENT** soft key



- ▶ Select the preset number that you want to activate



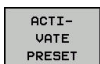
- ▶ Or, with the **GOTO** key, select the preset number that you want to activate



- ▶ Confirm with the **ENT** key



- ▶ Confirm with the **ENT** key



- ▶ Press the **ACTIVATE 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 in order to activate presets from the preset table during program run. In Cycle 247 you define the number of the preset to be activated.

Further information: Conversational Programming User's Manual

5.5 Presetting without a 3-D touch probe

Note

When presetting, you set the control display to the coordinates of a known workpiece position.



All manual probe functions are available with a 3-D touch probe.

Further information: "Presetting with a 3-D touch probe (option number 17)", Page 174



Refer to your machine manual.

The machine tool builder 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:



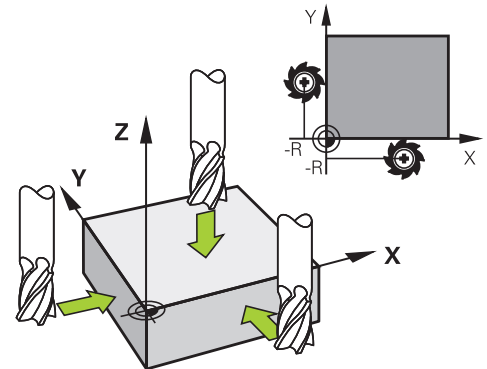
- Select the axis
- The control opens the **PRESETTING Z=** dialog window



- 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 tool builder has locked an axis, then you cannot set a preset in that axis. The soft key for that axis is then not visible.

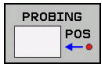
Using touch probe functions with mechanical probes or measuring dials

If you do not have an electronic 3-D 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 3-D touch probe (option 17)", Page 161

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key.

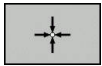
Proceed as follows:



- ▶ 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 166
- Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ 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 tool builder has defined.

5.6 Using a 3-D touch probe (option 17)

Overview


The following touch probe cycles are available in the **Manual operation** mode:








Refer to your machine manual.

The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

The touch probe cycles are available only with option number 17. If you are using a HEIDENHAIN touch probe, this option is available automatically.



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Function	Page
	Calibrating the 3-D Touch Probe	168
	Setting the preset on any axis	174
	Setting a circle center as preset	175
	Setting the centerline as preset	178
	Touch probe system data management	

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



If the handwheel is active, you cannot start the touch probe cycles.

Suppressing touch probe monitoring


Suppress touch probe monitoring

If the stylus is deflected, the control issues an error message as soon as you want to move a machine axis.

You must deactivate touch-probe monitoring in the **Manual operation** mode in order to use a positioning block to retract a touch probe after it has been deflected.

You can deactivate touch-probe monitoring for 30 seconds with the **TCH PROBE MONITOR OFF** soft key.

The control issues the error message **The touch probe monitor is deactivated for 30 seconds**. The error message automatically clears itself after 30 seconds.



If the touch probe receives a stable signal within the 30 seconds, such as "Touch probe not deflected," then touch-probe monitoring reactivates itself automatically and the error message is cleared.

NOTICE







Danger of collision!

The **TCH PROBE MONITOR OFF** soft key suppresses the corresponding error message if the stylus is deflected. The control does not perform an automatic collision check with the stylus. Because of this behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction.

► Carefully move the axes in the **Manual operation** 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
	Select the probing direction
	Capture the actual position
	Probe hole (inside circle) automatically
	Probe stud (outside circle) automatically
	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 model circles

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 pattern circle, 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:

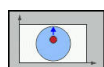
- ▶ Pre-position touch probe



- ▶ 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.



Operating and programming notes:

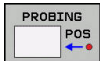
- Before starting an automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance opposite to the probing direction. The safety clearance is derived from the sum of the values in the touch-probe table and in 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.

Selecting the probing cycle

- ▶ Select the **Manual operation** or **Electronic handwheel** mode of operation



- ▶ Select the probing functions: Press the **TOUCH PROBE** soft key



- ▶ Select the touch probe cycle by pressing the appropriate soft key, for example **PROBING POS**
- ▶ The control displays the associated menu.



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 tool builder 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 in the **TNC:** main directory.



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




If you want to save measured values in the workpiece coordinate system, use the **ENTER 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 167

With the **ENTER IN DATUM TABLE** soft key, the control can write the values measured during any touch probe cycle to a datum table:

- ▶ Select any probe 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 **ENTER 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

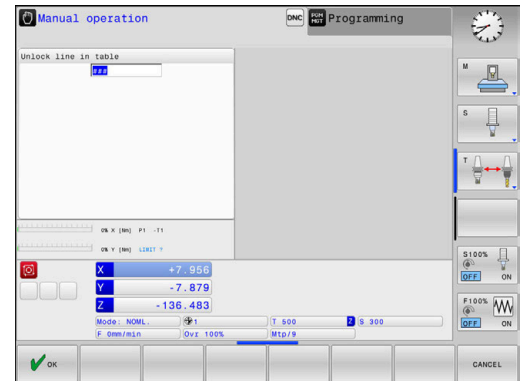
 If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function. If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function.

Further information: "Writing measured values from the touch probe cycles to a datum table", Page 166

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

i 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 3-D touch probes (option 17)

Introduction

In order to precisely specify the actual trigger point of a 3-D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.



Operating notes:

- Always calibrate the touch probe again in the following cases:
 - Initial configuration
 - Broken stylus
 - Stylus exchange
 - 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, there is no need to retrieve the tool again.

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D 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:



- ▶ Press the **TOUCH PROBE** soft key



- ▶ Display the calibration cycles: Press **CALIBRATE TS**
- ▶ Select the calibration cycle

Calibration cycles

Soft key	Function	Page
	Calibrating the length	169
	Measure the radius and the center offset using a calibration ring	170
	Measure the radius and the center offset using a stud or a calibration pin	170
	Measure the radius and the center offset using a calibration sphere	170

Calibrating the effective length

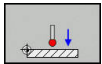


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

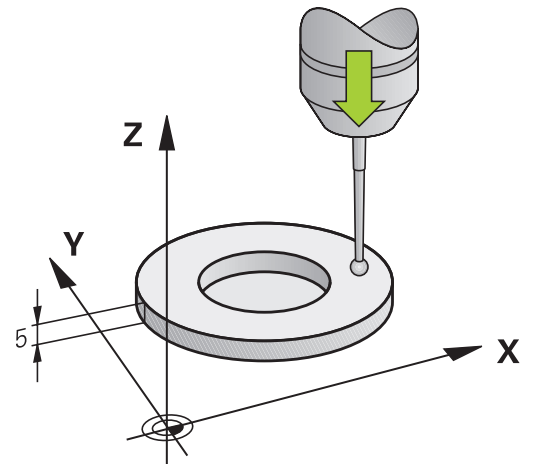


The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

- Set the preset in the spindle axis such that for the machine tool table Z=0.



- Select the calibration function for the touch probe length: Press the **CAL.** Press **L**
- 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



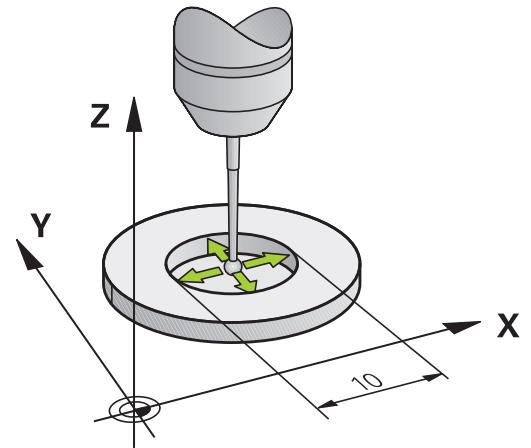
HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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 stud (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 tool builder.



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 preposition the touch probe above the center of the calibration sphere or calibration pin. Ensure that the touch points can be approached without collision.



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 calibration function: Press the **CAL. R** soft key
- ▶ The control displays the current calibration data.
- ▶ Enter the diameter of the ring gauge
- ▶ Probe: Press the **NC Start** key
- ▶ The 3-D 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.

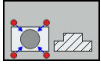


Refer to your machine manual.

In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.

Calibration 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 calibration function: Press the **CAL. R** soft key
- ▶ Enter the outside diameter of the stud
- ▶ Enter the safety clearance
- ▶ Probe: Press the **NC Start** key
- ▶ The 3-D 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.

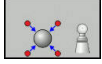


Refer to your machine manual.

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 calibration function: Press the **CAL. R** 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 3-D 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.

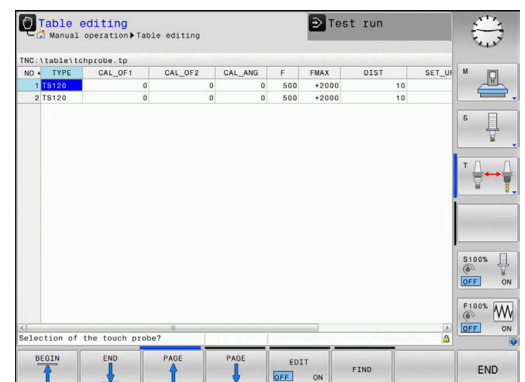
Displaying calibration values

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** (principal axis) and **CAL_OF2** (minor axis). You can display the values on the screen by pressing the **TCH PROBE TABLE** soft key.

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 3-D touch probe (option number 17)

Overview



Refer to your machine manual.




The machine tool builder 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 tool builder has defined.



HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

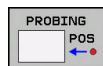
The following soft-key functions are available for setting a preset on an aligned workpiece:

Soft key	Function	Page
	Setting a preset in any axis	174
	Setting a circle center as preset	175
	Setting the center line as preset	178

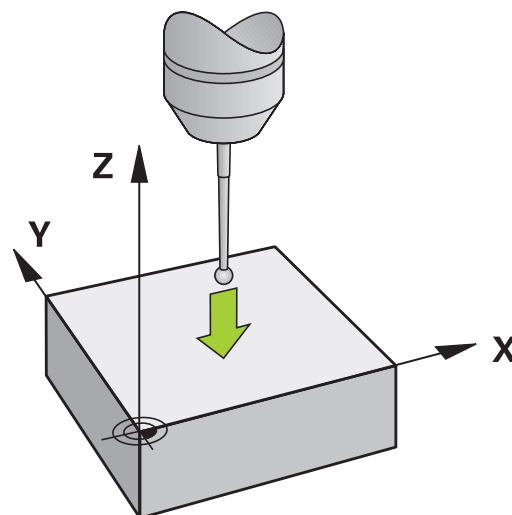


With an active datum shift the determined value is with respect to the current preset (possibly a manual preset from the **Manual operation** mode). The datum shift is taken into account in the position display.

Presetting on any axis



- ▶ 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
- ▶ Load with the **SET PRESET** soft key
- ▶ **Further information**: "Writing measured values from the touch probe cycles to a datum table", Page 166
- ▶ **Further information**: "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ 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.

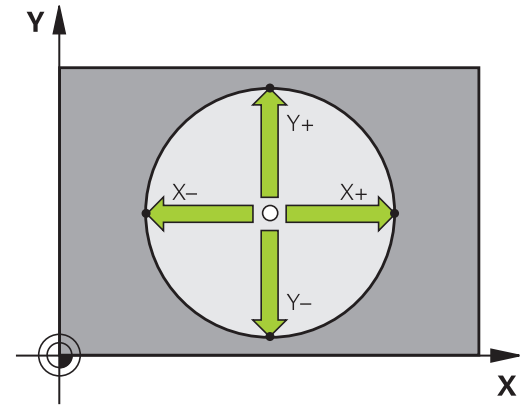
Inside circle:

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.



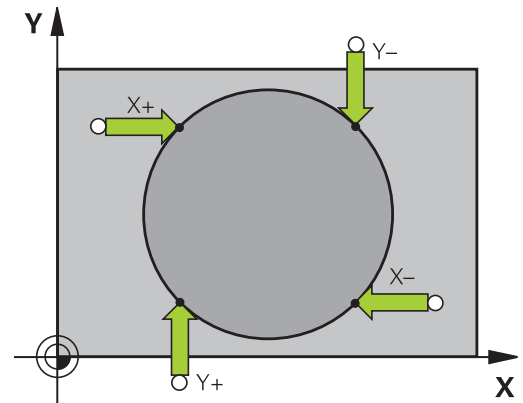
- ▶ Position the touch probe approximately in the center of the circle
- ▶ Select the probing function: Press the **PROBING CC** soft key
- ▶ Select the soft key for the desired probing direction
- ▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected 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 both coordinates of the center of the circle in the menu window
- ▶ Load with the **SET PRESET** soft key
- ▶ **Further information:** "Writing measured values from the touch probe cycles to a datum table", Page 166
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ 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
- ▶ Load with the **SET PRESET** soft key
- ▶ **Further information:** "Writing measured values from the touch probe cycles to a datum table", Page 166
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ 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 probing function **Probing of circular pattern** is part of the **Cir** probing function. Individual circles can be measured with paraxial probing operations.

A second soft-key row provides the **PROBING CC (probing of circular pattern)** soft key for using multiple holes or circular studs to set the preset. You can set the intersection of three or more elements as preset.

Setting the preset in the intersection of multiple holes or circular studs:

- ▶ Pre-position touch probe

Select the **Probing CC** probing function

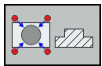


- ▶ Select the probing function: Press the **PROBING CC** soft key

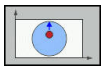


- ▶ Press the **PROBING CC (probing of circular pattern)** soft key

Probe a circular stud



- ▶ Circular stud should be probed automatically: Press **Stud** soft key

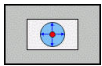


- ▶ Enter or select using soft key

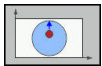


- ▶ Start probing function: Press the **NC start** key

Probe the hole.



- ▶ Hole should be probed automatically: Press the **Hole** soft key



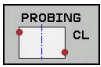
- ▶ Enter or select using soft key



- ▶ 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 both coordinates of the center of the circle in the menu window
- ▶ Load with the **SET PRESET** soft key
- ▶ **Further information:** "Writing measured values from the touch probe cycles to a datum table", Page 166
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ Terminate the probing function: Press the **END** soft key

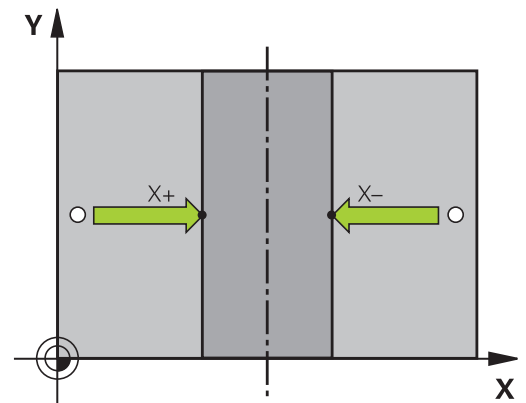
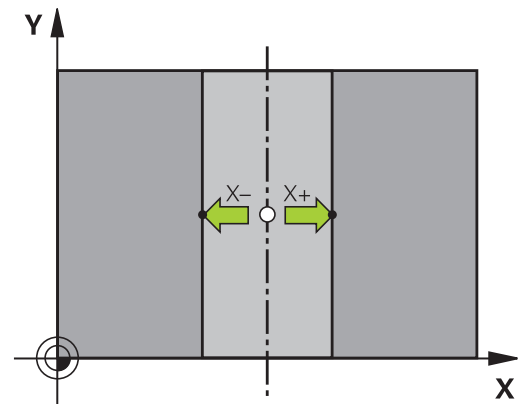
Setting a center line as preset



- ▶ 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 166
Further information: "Writing measured values from the touch probe cycles to the preset table", Page 167
- ▶ Terminate the probing function: Press the **END** soft key



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.



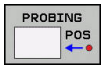
Measuring workpieces with a 3-D touch probe

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 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions on the workpiece

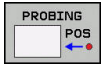
Finding the coordinates of a position on an aligned workpiece



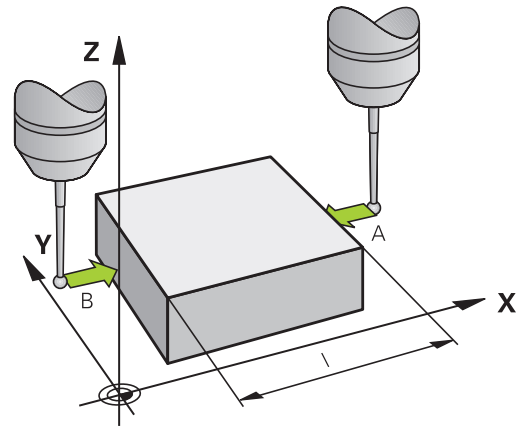
- ▶ Select the probing function: Press the **PROBING POS** soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the probing direction and the axis to which the coordinates relate: Use the corresponding soft keys to select
- ▶ Start the probing process: Press the **NC Start** 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.

To return to the values that were 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**



In **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.

For active tool tables, the control also takes the entries in the columns **L**, **LCUTS**, **T-ANGLE** and **R2** into consideration.

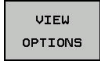
The control will not show a graphic if

- no NC program is selected
- the wrong screen layout is 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

Proceed as follows to access the **VIEW OPTIONS**:

- Select the desired operating mode
- 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** MOD function.

The control provides the following **VIEW OPTIONS**:

Soft keys	Function
	Show the workpiece
	Show the tool Further information: "Tool", Page 184
	Show the tool paths Further information: "Tool", Page 184
	Select the view Further information: "View", Page 185
	Reset the tool paths
	Reset the workpiece blank
	Show the workpiece blank frame
	Highlight the workpiece edges on 3-D model
	Show the block numbers of the tool paths
	Show the end points of the tool paths
	Show the workpiece in color
	Remove the chips Workpiece remainders (chips) after the milling process are removed.
	Reset the tool paths
	Rotate and zoom the workpiece Further information: "Rotating, zooming and moving a graphic", Page 186

Soft keys	Function
	Shift the sectional plane in the projection in three planes Further information: "Shifting the sectional plane", Page 188



Operating notes:

- With the machine parameter **clearPathAtBlk** (no. 124203), you specify whether the tool paths are cleared in the **Test Run** operating mode with a new BLK FORM.
- 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 **LCUT** columns are defined in the tool table, the tool is graphically displayed.




Further information: "Entering tool data into the table", Page 103

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

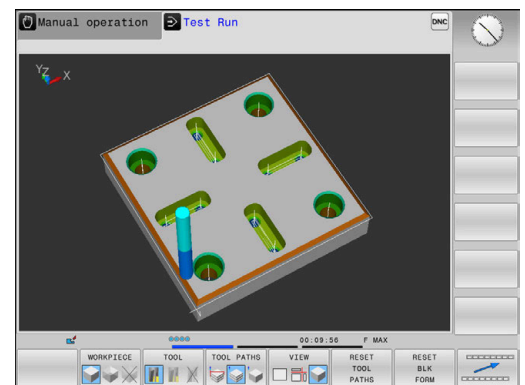
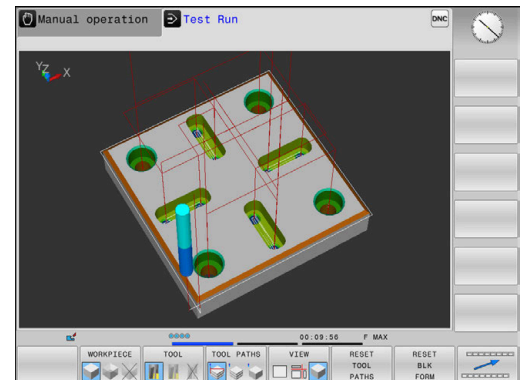
Showing the tool paths

The control displays the following types of movement:

Soft keys	Function
	Movements at rapid traverse and at the programmed feed rate
	Movements at the programmed feed rate
	No movements






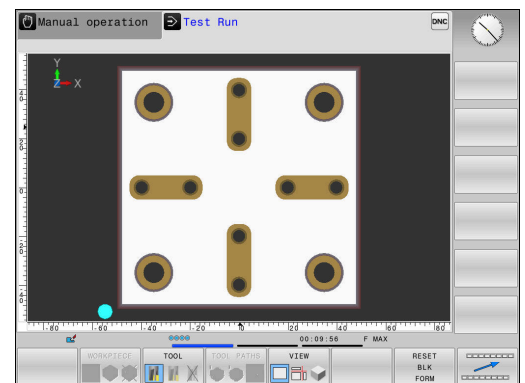
If you move at rapid traverse inside a workpiece, both the movement and the workpiece is shown in red at the respective location.



View

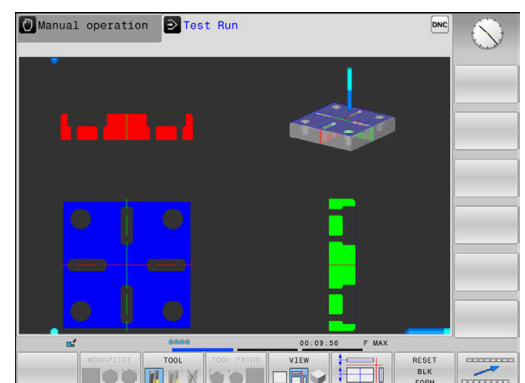
The control provides the following views:

Soft keys	Function
	Plan view
	Projection in three planes
	3-D view



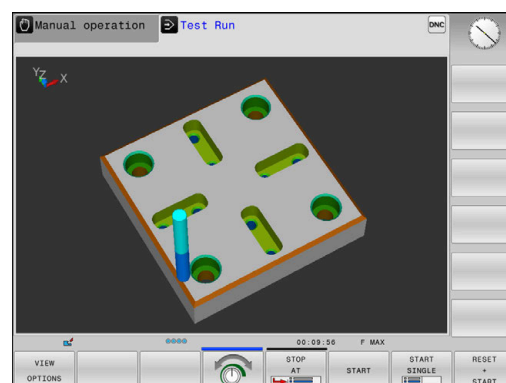
Projection in three planes

The simulation shows three sectional planes and a 3-D model, similar to a technical drawing.



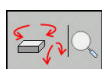
3-D view

The high-resolution 3-D 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 keys		Function
		Rotate in 5° steps about the vertical axis
		Tilt in 5° steps about the horizontal 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

Speed of the setting test runs





The most recently set speed stays active until a power interruption. After the control is switched on the speed is set to FMAX.

After you have started a program, the control displays the following soft keys with which you can set the simulation speed:



Soft key	Functions
	Test 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
	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

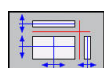
A part program can be graphically simulated as often as desired. To do so you can reset the graphic to the workpiece blank.

Soft key	Function
	Display the unmachined workpiece blank in the Program run, single block and Program run, full sequence operating modes
	Display the unmachined workpiece blank in the Test Run operating mode







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:

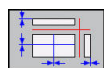
Soft keys	Function
 	Shift the vertical sectional plane to the right or left
 	Shift the vertical sectional plane forward or backward
 	Shift the horizontal sectional plane upwards or downwards

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 Showing the workpiece blank in the working space

Application

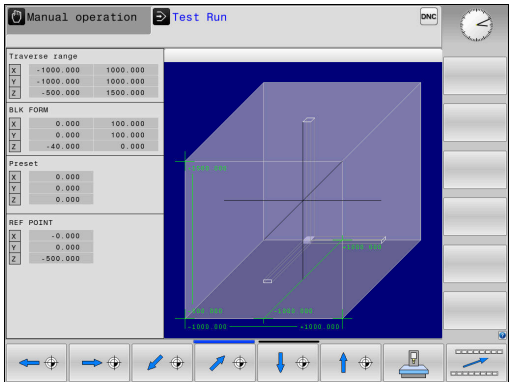
In **Test Run** operating mode, you can graphically check the position of the workpiece blank and the preset in the working space of the machine. The graphic shows the preset that has been set in the NC program using Cycle 247. If you have not defined a preset in the NC program, then the graphic shows the active preset on the machine.

You can activate working space monitoring in **Test Run** operating mode: Press the **BLANK IN WORK SPACE** soft key. You can activate or deactivate the function using the **SW limit monitoring** soft key.

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 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.

You can also activate the current preset for the **Test Run** operating mode.



Soft keys	Function
	Shift workpiece blank in positive/negative X direction
	Shift workpiece blank in positive/negative Y direction
	Shift workpiece blank in positive/negative Z direction
	Show workpiece blank referenced to the set preset
	Display the current traverse range
	The control shows the traverse ranges configured by the machine tool builder that can be selected as required.
	Switch monitoring function on or off
	Display machine reference point

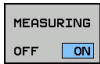
For blank in work space, the control depicts the **BLK FORM** only schematically.

- With **BLK FORM CYLINDER**, a cuboid is depicted as the workpiece blank

6.3 Measuring

Application

In **Test Run** operating mode, you can use the **MEASURING** soft key to show the coordinates.



- ▶ Set the **MEASURING** soft key to **ON**
- ▶ Position the mouse pointer on the respective position
- ▶ The control shows an approximated value for the corresponding coordinates.



The **MEASURING** soft key is available in the following views:

- Plan view
- 3-D view

Further information: "View", Page 185

6.4 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.

6.5 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**



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 prefaced with a slash (/):



- ▶ Set the **HIDE** soft key to **ON**
- > The control skips the NC blocks.



- ▶ Set the **HIDE** soft key to **OFF**
- > The control runs or tests the NC blocks.

Procedure

You can optionally hide NC blocks.

Proceed as follows to hide NC blocks in the **Programming** operating mode:



- ▶ Select the desired NC block



- ▶ Press the **INSERT** soft key
- > The control inserts a slash (/).

Proceed as follows to show NC blocks in the **Programming** operating mode again:



- ▶ Select the hidden NC block



- ▶ 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:



- ▶ 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

Proceed as follows 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 (**/**).

Proceed as follows 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 (**/**).

6.6 Test run

Application

In the **Test Run** operating mode, you can simulate NC programs and program sections in order to reduce programming errors during program run. The control checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space
- Using disabled tools

The following functions are also available:

- Blockwise test run
- Interruption of test at any NC block
- Skip NC blocks
- Functions for graphic simulation
- Measure machining time
- Additional status display

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**

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). This can cause a test performed without errors to later deviate from the machining operation. 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 repositioning
- ▶ Carefully test the NC program in the **Program run, single block** operating mode




Refer to your machine manual.

Your machine tool builder 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 tool builder 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.






After **RESET + START**, line 0 of the temporarily loaded preset table automatically displays the currently active preset from **Preset.PR** (execution). Line 0 is selected when starting the test run until you define another preset in the NC program. All presets from lines > 0 are read by the control from the selected preset table of 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 189

- 
- ▶ Operating mode: Press the **Test Run** key
- 
- ▶ 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	Functions
	Reset the blank form, reset the previous tool data and test the entire NC program
	Test the entire NC program
	Test each NC block individually
	Executes the Test Run until NC block N
	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 fixed 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

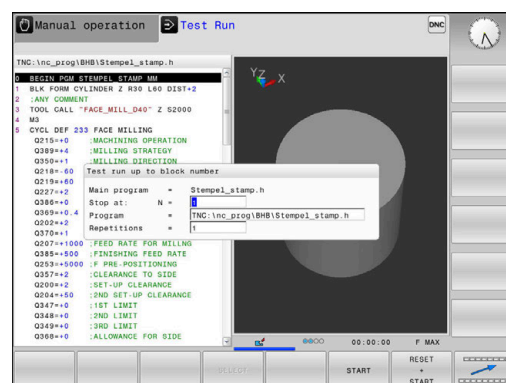
Executing a Test Run up to a certain NC block

With the **STOP AT** function the control executes a **Test Run** up to the NC block with block number **N**.

Proceed as follows to stop the **Test Run** at any 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 **PGM CALL**, 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**



GOTO function

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:

- 
- ▶ Press the **GOTO** key
 - > The control opens a pop-up window.
 - ▶ Enter a number
- 
- ▶ 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
	Move up the number of lines entered
	Move down the number of lines entered
	Jump to the block number entered





Use the **GOTO** function only during programming and testing of NC programs. Use the block scan function during program run.

Further information: "Any entry into the NC program: Block scan", Page 212

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.

Proceed as follows to select special functions:

- 
- ▶ Press the **SPEC FCT** key
- 
- ▶ 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 Conversational 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.

Display of NC programs

Syntax highlighting

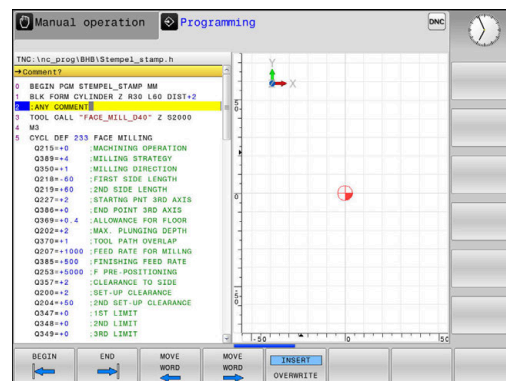
The control displays syntax elements with various colors according to their meaning. Color-highlighting makes the NC programs easier to read and clearer.

Color highlighting of syntax elements

Use	Color
Standard color	Black
Display of comments	Green
Display of numerical values	Blue
Display of the block number	Violet
Display of FMAX	Orange
Display of the feed rate	Brown

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.



6.7 Program run

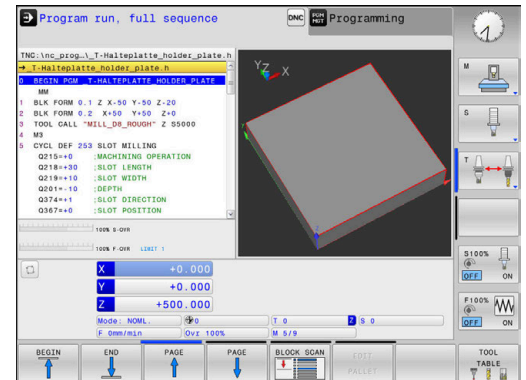
Application

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.

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
- Checking and changing Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display



Running an NC program

Preparation

- 1 Clamp the workpiece to the machine table.
- 2 Set the preset
- 3 Select the pallet files (status M)
- 4 Select the NC program (status M)



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.

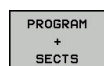
Structure blocks can also be displayed in a separate window, and be edited or added to, as desired. Use the appropriate screen layout for this.

The control manages the inserted structure items in a separate file (extension: .SEC.DEF). 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



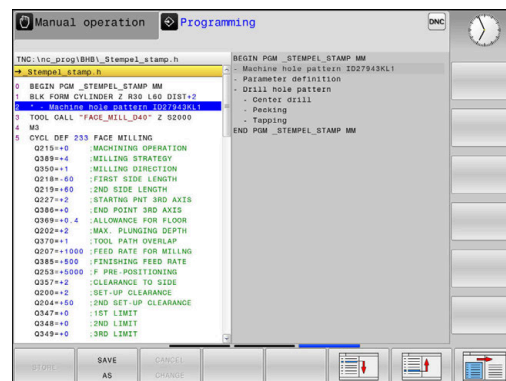
- ▶ Display structure window: For this screen layout press the **PROGRAM + SECTS** soft key



- ▶ 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

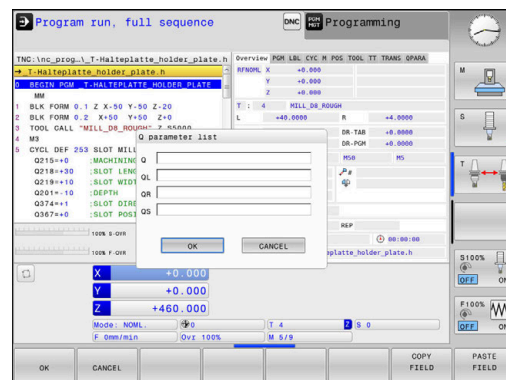
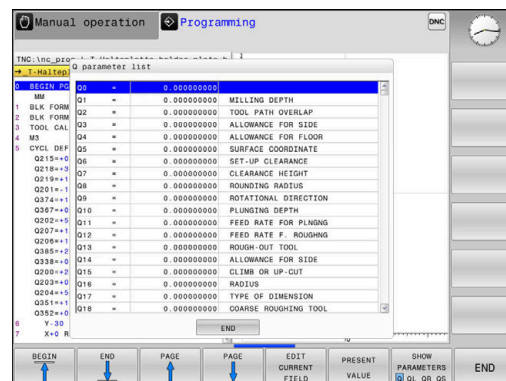
Q
INFO

- ▶ 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 would like to change the value, press the **EDIT CURRENT FIELD** soft key. Enter a new value and confirm with the **ENT** key
- ▶ To leave the value unchanged, press the **PRESENT VALUE** soft key or close the dialog with the **END** key



All of the parameters with displayed comments are used by the control within cycles or as transfer parameters.

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.



You can have Q parameters also displayed in the additional status display in all operating modes (except **Programming** mode).

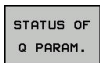
- ▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC stop** key and the **INTERNAL STOP** soft key), or stop the test run



- ▶ Call the soft key row for screen layout



- ▶ Select the layout option for the additional status display
- In the right half of the screen, the control shows the **Overview** status form.



- ▶ Press the **STATUS OF Q PARAM.** soft key



- ▶ 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 control. 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, whereby e-08 corresponds to the factor of 10^{-8} .

Interrupting, stopping or aborting machining

There are several ways to stop a program run:

- Interrupt the program run with e.g. the miscellaneous function **M0**
- Interrupt the program run e.g. with the miscellaneous function **M0**
- Stop the program run, e.g. with the **NC stop** key in connection with the **INTERNAL STOP** soft key
- Terminate the program run e.g. with the miscellaneous functions **M2** or **M30**

The control shows the current status of the program run in the status display.

Further information: "General status display", Page 54

In contrast to a stopped run, an interrupted, aborted (terminated) program run enables certain actions by the user, including the following:

- Select operating mode
- Check Q parameters and change these if necessary using the **Q INFO** function
- Change setting for the optional programmed interruption with **M1**
- Change setting for the programmed skipping of NC blocks with **/**



During major errors, the control automatically aborts the 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 cause the control to lose program information affecting the mode and thereby to lose the so-called contextual reference. After the loss of the contextual reference, unexpected and undesired movements can occur. There is a danger of collision during subsequent machining operations!

- ▶ 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 Q parameter values with the **Q INFO** soft key
 - Switching the operating modes
- ▶ Restore the contextual reference via repetition of the required NC blocks



Refer to your machine manual.

The miscellaneous function **M6** may also lead to a suspension of the program run. The machine manufacturer sets the functional scope of the miscellaneous functions.

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.

Moving the machine axes during an interruption

You can move the machine axes during an interruption 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).



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 tool builder configures and enables the **Retract** operating mode.

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
- Traverse 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 unable to reset them manually.

NOTICE

Caution: Danger to the tool and workpiece!

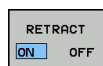
A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last 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.



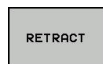
- ▶ Activate the **Retraction** mode: Press the **RETRACT** soft key
- > The control displays the message **Retraction selected**



- ▶ 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

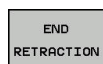


- ▶ Activate retraction: Press the **RETRACT** soft key

- ▶ 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 the **Retraction** mode: Press the **END RETRACTION** soft key
- > The control checks whether the **Retraction** 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, cross the reference points
- ▶ Establish the desired machine condition

Any entry into the NC program: Block scan



Refer to your machine manual.

The **BLOCK SCAN** function must be enabled and configured by the machine tool manufacturer.

With the **BLOCK SCAN** function you can start an NC program at any desired NC block. The control takes workpiece machining up to this NC block into account in its calculations.

If the NC program was interrupted under the following conditions, the control saves the interruption point:

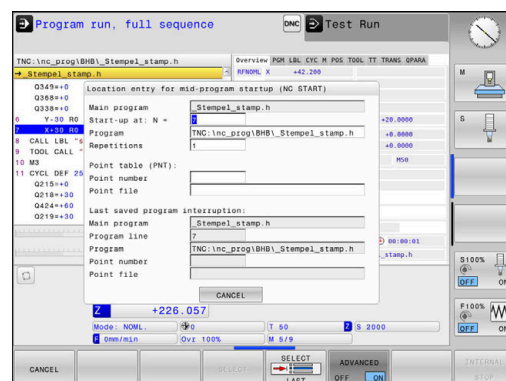
- **INTERNAL STOP** soft key
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute mid-program startup directly at the point of interruption.

You can run the mid-program startup in the following ways:

- Mid-program startup in the main program, with repetitions if necessary
- Multi-level mid-program startup in subprograms and touch probe cycles
- Mid-program startup in a point table
- Block scan in pallet programs

At the start of the block scan, the control resets all data, as with a selection of the 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 at multiple levels

Further information: "Procedure for multi-level mid-program startup", Page 214

Procedure for simple mid-program startup

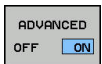


The control only displays the dialogs required by the process in the pop-up window.



- ▶ 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:** Enter the number of repetitions which should be taken into account in the block scan if the NC block is located within a program section repeat.

Default 1 means first machining



- ▶ Press the **ADVANCED** soft key if required



- ▶ if required, press the **INSERT LAST NC BLOCK** soft key 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, 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 217



- ▶ Press the **NC Start** key
- The control resumes execution of the NC program.

Example of simple mid-program startup

After an internal stop, you would like to start in NC block 12 in the third machining operation of LBL 1.

In the pop-up window enter the following data:

- **Start-up at: N =12**
- **Repetitions 3**

Procedure for multi-level mid-program startup

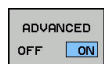
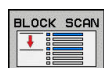
If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this purpose, jump in the main program to the desired subprogram call. With the **CONTINUE BLOCK SCAN** function, you can jump further from this position.



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.

Mid-program startup 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 required
- ▶ If required, press the **INSERT LAST NC 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.

Mid-program startup 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



- ▶ 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

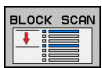


- ▶ Press the **NC Start** key
- ▶ The control resumes execution of the NC program.

Example of multi-level mid-program startup

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.



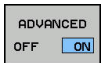
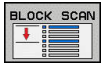
- ▶ Press the **CONTINUE BLOCK SCAN** soft key
- ▶ In the pop-up window enter the following data:
 - **Start-up at: N =8**
 - **Repetitions 1**



- ▶ 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.



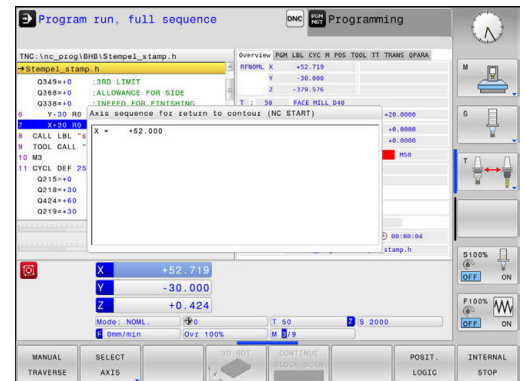
- ▶ Press the **BLOCK SCAN** soft key
- > The control shows a pop-up window.
- ▶ Press the **ADVANCED** soft key
- > The control expands the pop-up window.
- ▶ **Point number:** Enter the line number of the point table you start with
- ▶ **Point file:** Enter the name and path of the point table
- ▶ If required, press the **SELECT LAST BLOCK** soft key in order to select the last saved interruption
- ▶ 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

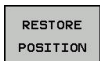
With the **RESTORE POSITION** function, the control moves the tool to the workpiece contour in the following situations:

- 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 with a block scan with **RESTORE POS AT N**, for example 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

Proceed as follows to approach the contour:



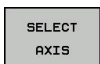
- ▶ Press the **RESTORE POSITION** soft key
- ▶ Restore the machine status, if required

Approach the axes in the sequence shown by the control:



- Press the **NC Start** key

Approach the axes according to individually selected sequence:



- ▶ Press the **SELECT AXIS** soft key
- ▶ Press the axis soft key of the first axis
- ▶ Press the **NC Start** key



- ▶ Press the axis soft key of the second axis
- ▶ Press the **NC Start** key



- ▶ Repeat the process for all axes







If the tool is located in the tool axis below the starting point, then the control offers the tool axis as the first traverse direction.

6.8 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	Functions
	Go back one screen in the NC program
	Go forward one screen in the NC - program
	Select start of program
	Select end of program

6.9 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 conversational programming and execute it immediately. The NC program is stored in the file \$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 219



Operating and programming notes:

- The following functions are not available in the **Positioning w/ Manual Data Input** operating mode:
 - Program call
 - **PGM CALL**
 - **SEL PGM**
 - **CALL SELECTED PGM**
 - Programming graphics
 - Program-run graphics
- Using the **SELECT BLOCK** and **CUT OUT BLOCK** soft keys etc. you can also conveniently and rapidly reuse program sections from other NC programs.

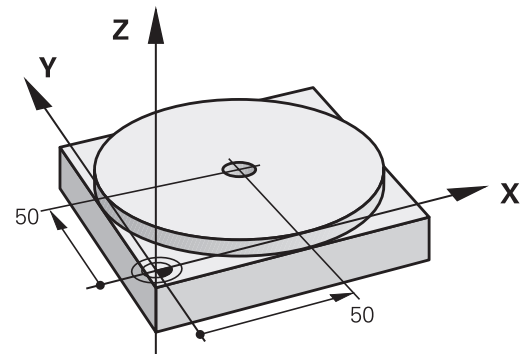
Further information: Conversational Programming User's Manual
- You can control and modify Q parameters with the soft keys **Q PARAMETER LIST** and **Q INFO**.

Further information: "Checking and changing Q parameters", Page 202

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**.



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 DRILLING 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:



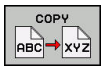
- ▶ Operating mode: Press the **Programming** key



- ▶ To call the file manager, press the **PGM MGT** key.



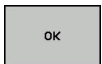
- ▶ Move the highlight to the **\$MDI** file



- ▶ To 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**



- ▶ Press the **OK** soft key.



- ▶ Exit the file manager: Press the **END** soft key

6.10 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. Some miscellaneous functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the **Manual operation** and **Electronic handwheel** operating modes, the M functions are entered with the **M** soft key.

Effectiveness of miscellaneous functions

Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

Miscellaneous functions come into effect in the NC block in which they are called.

Some miscellaneous functions are effective only in the NC block in which they are programmed. Unless the miscellaneous function is only effective blockwise, you must either cancel it in a subsequent NC block with a separate M function, or it is automatically canceled by the control at the end of the program.



If multiple 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

6.11 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.

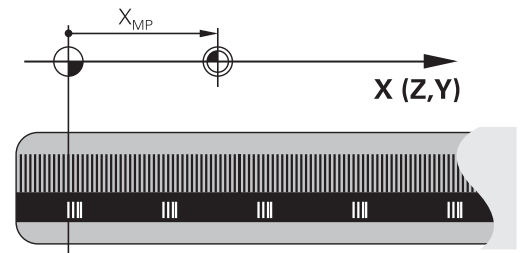
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope depends on machine parameter resetAt (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M6	Tool change Spindle STOP Program STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

6.12 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 (e.g. 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 3-D touch probe", Page 158

Behavior with M91 – Machine datum

If you want the coordinates in a positioning block to be based on the machine datum, enter M91 into these NC blocks.



If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the control screen reference the machine datum. Switch the display of coordinates in the status display to REF.

Further information: "Status displays", Page 54

Behavior with M92 – Additional machine reference point



Refer to your machine manual.

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a machine reference point.

For each axis, the machine tool builder defines the distance between the machine reference point 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.

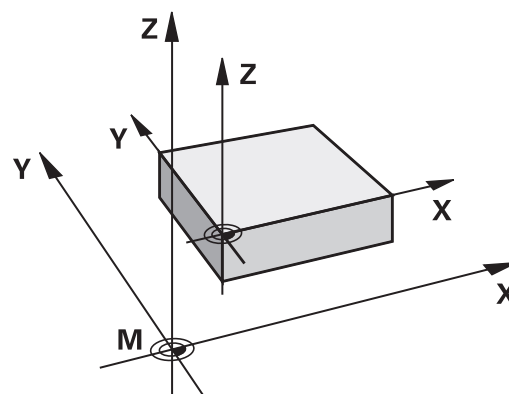
M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always be referenced to the machine datum, you can disable the setting of presets for one or more axes.

If presetting is inhibited for all axes, the control no longer displays the **SET PRESET** soft key in the **Manual operation** 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 189

7

Special Functions

7.1 Defining a counter

Application



Refer to your machine manual.
Your machine manufacturer enables this function.

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Proceed as follows for the definition:

SPEC
FCT

- ▶ Show the soft key row with 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 count you have defined directly in the NC program is effective. The count in the MOD menu remains unaffected.

Effect in the Program Run Single Block and Program Run Full Sequence operating modes

The count from the MOD menu is only effective in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

The count is retained even after a restart of the control.

Define FUNCTION COUNT

The **FUNCTION COUNT** function provides the following possibilities:

Soft key	Meaning
FUNCTION COUNT INC	Increase count by 1
FUNCTION COUNT RESET	Reset counter
FUNCTION COUNT TARGET	Set the nominal count (target value) to the desired value Input value: 0–9999
FUNCTION COUNT SET	Set the counter to the desired value Input value: 0–9999
FUNCTION COUNT ADD	Increment the counter by the desired value Input value: 0–9999
FUNCTION COUNT REPEAT	Repeat the NC program starting from this label if more parts are to be machined.

Example

5 FUNCTION COUNT RESET	Reset the counter value
6 FUNCTION COUNT TARGET10	Enter the target number of parts to be machined
7 LBL 11	Enter the jump label
8 ...	Machining
51 FUNCTION COUNT INC	Increment the counter value
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

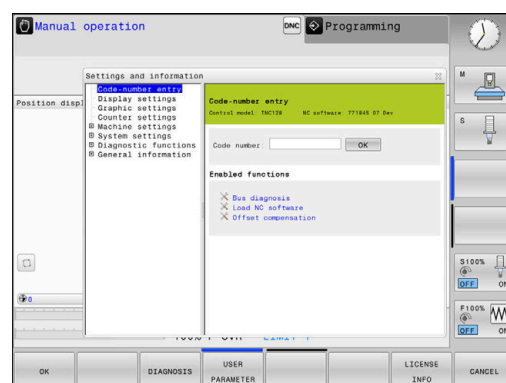
8.1 MOD function

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

Open the pop-up window with the MOD functions:

- ▶ 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

- ▶ Exit the MOD functions: Press the **CANCEL** soft key or the **END** key

Overview of MOD functions

The following functions are available independent of the selected operating mode:

Code-number entry

- Code number

Display settings

- Position displays
- Unit of measure (mm/inch) for position display
- Program input 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

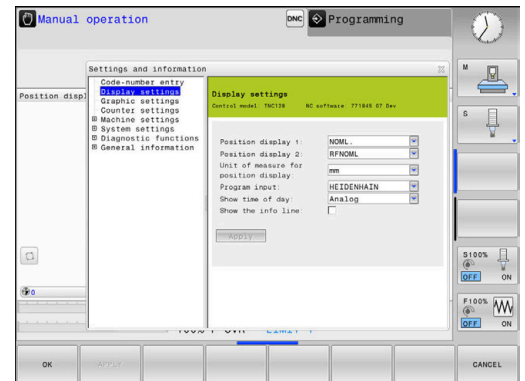
- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- HEROS information

General information

- Version information
- License information
- Machine times



8.2 Display software numbers

Application

The following software numbers are displayed on the control's screen after the **Software version** MOD function has been selected:

- **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:** Number or name of the PLC software (managed by your machine manufacturer)

In the **FCL Information** MOD function, the control shows the following information:

- **Development level (FCL=Feature Content Level):**
Development level of the software installed on the control
Further information: "Feature Content Level (upgrade functions)", Page 26

8.3 Entering the code number

Application

The control requires a code number for the following functions:

Function	Code number
Select user parameters	123
Configuring an Ethernet card	NET123
Enabling special functions for Q parameter programming	555343

Functions provided to the machine tool builder 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 tool builder to install software updates on the control.

NOTICE

Caution: Possible loss of data!

An incorrect procedure when installing updates can cause a loss of data.

Do not install any software updates without instructions!

Contact your machine manufacturer.

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. The files are thus permanently gone.

- ▶ Perform a backup of the current machine configuration prior to the **RESTORE** function
- ▶ Use the function only in consultation with the machine tool builder

Your machine tool builder can provide you a backup with a machine configuration. After entering the keyword **RESTORE**, you can load the backup on your machine or programming station. Proceed as follows to load the backup:

- ▶ Enter the keyword **RESTORE** in the MOD dialog
- ▶ 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 to start the backup process

8.5 Selecting the position display

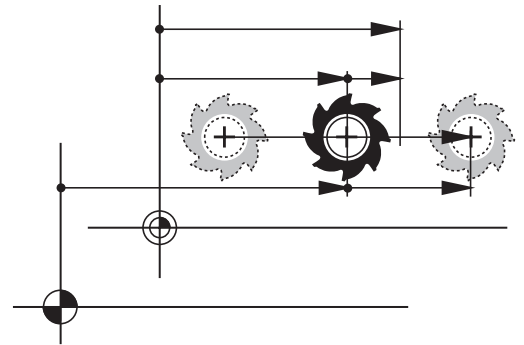
Application



You can influence the display of the coordinates for the **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
	<div>  The NOML and ACTL displays differ solely with regard to following error. </div>
ACTL	Actual position; current tool position
	<div>  Refer to your machine manual. Your machine tool builder defines whether the ACTL and NOML display deviates from the programmed position by the DL oversize of the tool call. </div>
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: <ul style="list-style-type: none"> ▶ Scaling factor 0.2 ▶ L IX+10 > The ACTDST display shows 10 mm. > The scaling factor does not have any influence.

Display	Function
REFDST	<p>Distance remaining to the programmed position in the machine coordinate system; difference between actual and target positions</p> <p>Examples with Cycle 11</p> <ul style="list-style-type: none"> ▶ 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	<p>Traverse paths that were executed with handwheel superimpositioning function (M118)</p>

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

With this MOD function, you can determine whether the control coordinates are displayed in millimeters 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), value is displayed to 4 decimal places

If you would like to activate the inch display, the control shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

8.7 Graphic settings

With the **Graphic settings** MOD function, you can select the model type and model quality for the **Test Run** operating mode.




Proceed as follows to select **Graphic settings**:

- ▶ Select the **Graphic settings** group in the MOD menu
- ▶ Select the model type
- ▶ Select the model quality
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key.


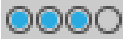


In the **Test Run** operating mode, the control displays icons of the active **Graphic settings**.

You have the following simulation parameters for the control's **Graphic settings**:

Model type

Icon	Choice	Properties	Application
	3-D	Very true to detail, heavy time and processor consumption	Milling with undercuts,
	2.5 D	Fast	Milling without undercuts
	No model	Very fast	Line graphics

Model quality

Icon	Choice	Properties
	Very high	High data transfer rate, exact depiction of tool geometry, depiction of block end points and block numbers possible
	High	High data transfer rate, exact depiction of tool geometry
	Medium	Medium data transfer rate, approximation of tool geometry
	Low	Low data transfer rate, coarse approximation of tool geometry

8.8 Setting the counter




With the **Counter settings** MOD function, you can change the current count (actual value) and the target value (nominal value).

Proceed as follows to select **Counter settings**:

- ▶ Select the **Counter settings** group in the MOD menu
- ▶ Select the current count
- ▶ Select the target value for the counter
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key

The control immediately takes over the selected value in the status display

Proceed as follows to change the **Counter settings** by soft key:

Soft key	Meaning
	Reset count
	Increase count
	Lower count

You can also enter the values directly with a connected mouse.

Further information: "Defining a counter", Page 228

8.9 Changing the machine settings

Selecting the kinematics



Refer to your machine manual.
Your machine tool builder configures and enables the **Kinematics selection** function.

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 selection** function only in the **Test Run** operating mode
- ▶ Use the **Kinematics selection** function for selecting the active machine kinematics only as needed

You can use this function to test NC programs whose kinematics does not match the active machine kinematics. If your machine tool builder saved different kinematic models in your machine and enabled them for selection, you can activate one of these kinematics configurations with the MOD function. When you select a kinematics model for the test run this does not affect machine kinematics.



Ensure that you have selected the correct kinematics in the Test Run operating mode for checking your workpiece.

Entering traverse limits

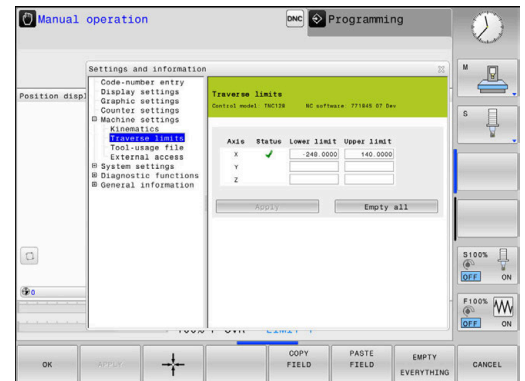


Refer to your machine manual.
Your machine tool builder configures and enables the **Traverse limits** function.

The **Traverse limits** MOD function enables you to limit the actually usable traverse path within the maximum range of traverse. This enables you to define protection zones on each axis in order, for example, to protect an indexing head from collision.

To enter traverse limits:

- ▶ In the MOD menu select the **Machine settings** group
- ▶ Select the **Traverse limits** menu
- ▶ Enter the values of the desired axes as a reference value or load the momentary position with the **actual position capture** soft key
- ▶ Press the **APPLY** soft key
- ▶ The control checks the entered values for validity.
- ▶ Press the soft key **OK**



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.
- You can only deactivate the protection zone by deleting all values or pressing the **EMPTY EVERYTHING** soft key.

Generating a tool usage file



Refer to your machine manual.

The tool usage test function must be enabled by your machine tool builder.

With the **Tool-usage file** MOD function, you can select whether the control never, once, or always creates a tool usage file.

Generate a tool usage file:

- ▶ In the MOD menu, select the **Machine settings** group
- ▶ Select the **Tool-usage file** menu
- ▶ Select the desired setting for the **Program Run, Full Sequence/Single Block** and **Test Run** operating modes
- ▶ Press the **APPLY** soft key
- ▶ Press the **OK** soft key

Permitting or restricting external access



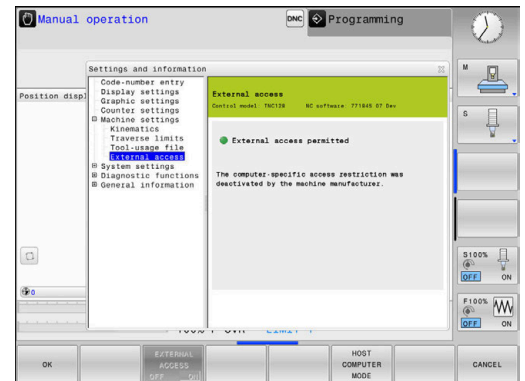
Refer to your machine manual.

The machine tool builder can configure the external access options.

With the **External access** MOD function, you can grant or restrict access to the control. Once you have restricted external access, it is no longer possible to connect to the control and to exchange data over a network or over a serial connection (e.g., with the **TNCremo** software).

Proceed as follows to restrict external access:

- ▶ In the MOD menu, select the **Machine settings** group
- ▶ Select the **External access** menu
- ▶ Set the **EXTERNAL ACCESS ON/OFF** soft key to **OFF**
- ▶ Press the **OK** soft key



Computer-specific access control

If your machine tool builder has set up computer-specific access control (machine parameter **CfgAccessControl** no. 123400), you can permit access for up to 32 connections authorized by you.

Proceed as follows:

- ▶ Select **Add** to create a new connection
- > 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 (text is shown in the overview list)

Type:

Ethernet	Network connection
Com 1	Serial interface 1
COM 2	Serial interface 2

Access rights:

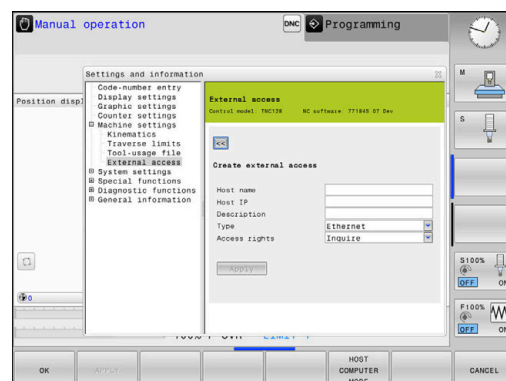
Inquire	For external access, the control opens a query dialog
Deny	Do not permit network access
Permit	Permit network access without query

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 permit or deny external access in the pop-up window:

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 tool builder.

With the **HOST COMPUTER MODE** soft key you transfer the command to an external host computer in order to transfer data to the control, for example.

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

Proceed as follows to start the host computer mode:

- ▶ In the MOD menu, select the **Machine settings** group
- ▶ Select the **External access** menu
- ▶ Press the **HOST COMPUTER MODE** soft key
- > The control shows an empty screen with the **Host computer is active** pop-up window.



Your machine tool builder can specify that the host computer mode can automatically be activated externally.

Exit the host computer mode as follows:

- ▶ Press the **HOST COMPUTER MODE** soft key again

8.10 Configuring the HR 550FS radio handwheel

Application



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.

To configure the HR 550FS radio handwheel, use the **SET UP WIRELESS HANDWHEEL** soft key. 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



Any changes or modifications that were not expressly approved by the party responsible for conformity could void your authority to operate the device.

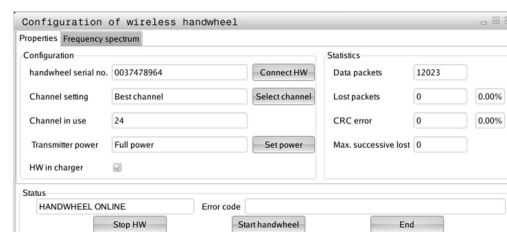
This device complies with Part 15 of the FCC Rules and the RSS standard(s) of Industry Canada for license-exempt equipment.

Operation is subject to the following conditions:

- 1 This device must not cause any harmful interference, and
- 2 this device must accept any interference received, including interference that may cause undesired operation.

Assigning the handwheel to a specific handwheel holder

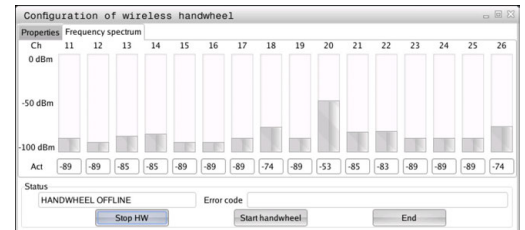
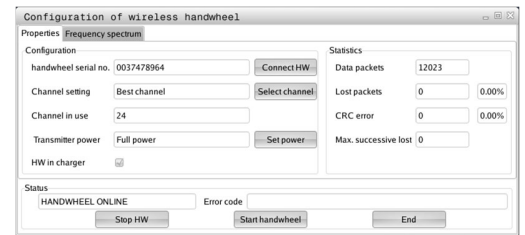
- ▶ Make sure that the handwheel holder is connected to the control hardware.
- ▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click on the **Connect HW** button
- ▶ The control saves serial number of the inserted radio handwheel and shows it in the configuration window on the left next to the **Connect HW** button.
- ▶ Save the configuration and close the configuration menu: Press the **END** button



Setting the transmission channel

If the wireless handwheel is started automatically, then the control tries to select the transmission channel providing the best transmission signal. Proceed as follows if you want to set the radio channel yourself:

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Frequency spectrum** tab
- ▶ Click the **Stop HW** button
- ▶ The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- ▶ Memorize the number of the channel with the least amount of radio traffic (smallest bar)
- ▶ Click the **Start handwheel** button to reactivate the radio handwheel
- ▶ Click the **Properties** tab
- ▶ Click the **Select channel** button
- ▶ The controls shows all available channel numbers
- ▶ Click the number of the channel that the control has found to have the least amount of radio traffic
- ▶ To save the configuration and exit the configuration menu, press the **END** button

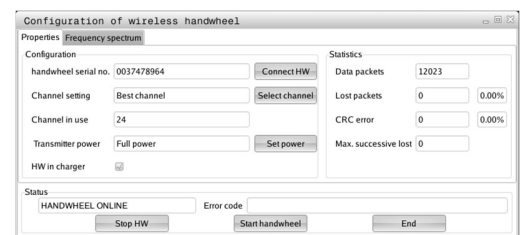


Selecting the transmitter power



A reduction in transmission power decreases the range of the wireless handwheel.

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Set power** button
- ▶ The control displays the three available power settings. Click on the desired setting.
- ▶ To save the configuration and exit the configuration menu, press the **END** button



Statistical data

To display the statistical data, proceed as follows:

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ The control displays the configuration menu with the statistical data.

Under **Statistics**, the control displays information about the transmission quality.

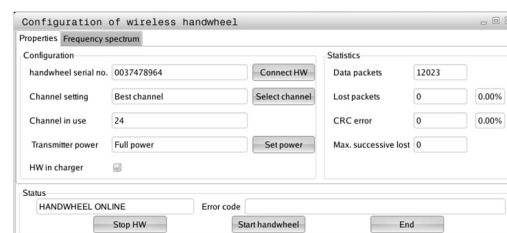
If the reception quality is poor so that a proper and safe stop of the axes cannot be ensured anymore, an emergency-stop reaction of the wireless handwheel is triggered.

The displayed value **Max. successive lost** indicates whether reception quality is poor. If the control repeatedly displays values greater than 2 during normal operation of the radio handwheel within the desired range of use, then there is a high risk of an undesired disconnection. This can be corrected by increasing the transmitter power or by changing to another channel with less radio traffic.

If this occurs, try to improve the transmission quality by selecting another channel or by increasing the transmitter power.

Further information: "Setting the transmission channel", Page 249

Further information: "Selecting the transmitter power", Page 249



8.11 Changing the system settings

Set the system time

With the **Set the system time** MOD function you can set the time zone, date and time manually or by means of an NTP server synchronization.

Proceed as follows to set the system time:

- ▶ In the MOD menu, select the **System settings** group
- ▶ Press the **SET DATE/ TIME** soft key
- ▶ In the **Time zone** area, select the desired time zone
- ▶ Press the **NTP on** soft key in order to select the **Set the time manually** entry
- ▶ Change the date and time as needed
- ▶ Press the **OK** soft key

To set the system time with the aid of an NTP server:


- ▶ In the MOD menu, select the **System settings** group
- ▶ Press the **SET DATE/ TIME** soft key
- ▶ In the **Time zone** area, select the desired time zone
- ▶ Press the **NTP off** soft key in order to select the **Synchronize the time over NTP server** entry
- ▶ Enter hostnames or the URL of an TNP server
- ▶ Press the **Add** soft key
- ▶ Press the **OK** soft key

8.12 Displaying operating times

Application

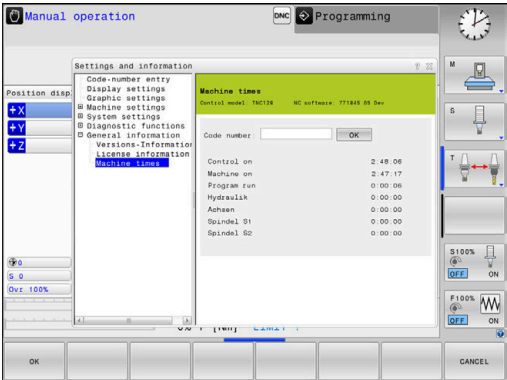
The **MACHINE TIME** MOD function enables you to see various types of operating times:

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 tool builder can provide further operating time displays.



9

HEROS functions

9.1 Window manager



Refer to your machine manual.

The machine tool builder 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 task bar 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



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.

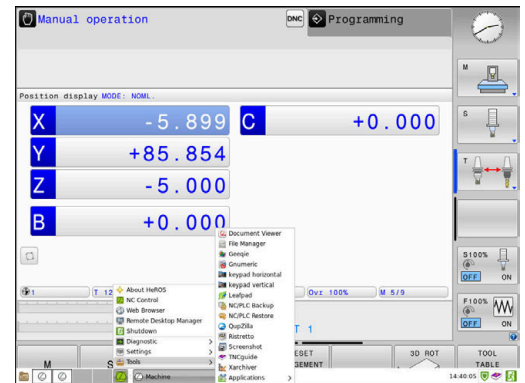
Overview of taskbar

In the task bar you can choose different workspaces by mouse click.

The control provides the following workspaces:

- Workspace 1: Active operating mode
- Workspace 2: Active programming mode
- Workspace 3: or applications of the machine tool builder (optionally available)
- Workspace 4: applications of the machine tool builder (optionally available)

In addition, you can also select other applications from the task bar which you have started in parallel to the control software, e.g. the **TNCguide**.



You can randomly move all open applications to the right of the green HEIDENHAIN symbol between the workspaces by pressing and holding the left mouse button.

Click the green HEIDENHAIN symbol to open a menu in which you can get information, make settings or start applications.

The following functions are available:

- **About HEROS:** Open information about the operating system of the control
- **NC Control:** Start and stop the control software (for diagnostic purposes only)
- **Web Browser:** Start the web browser
- **Diagnostic:** Diagnostic applications
 - **GSmartControl:** Available only to authorized specialists
 - **HE Logging:** Define settings for internal diagnostic files
 - **HE Menu:** Available only to authorized specialists
 - **perf2:** Check processor load and process load
 - **Portscan:** Test active connections
Further information: "Portscan", Page 258
 - **Portscan OEM:** Available only to authorized specialists
 - **RemoteService:** Start and stop remote maintenance
Further information: "Remote Service", Page 259
 - **Terminal:** Enter and execute console commands
- **Settings:** Operating system settings

- **Date/Time:** Set date and time
- **Language/Keyboards:** Select system dialog language and keyboard version—the control overwrites the setting of the system dialog language when starting with the language setting of the machine parameter **CfgDisplayLanguage** (no. 101300)
- **Network:** Define network settings
- **Printer:** Configure and manage printer
Further information: "Printer", Page 261
- **Screensaver:** Setting the screensaver
Further information: "Screensaver with lockout", Page 316
- **Current User:** Show the current user
Further information: "Current User", Page 318
- **UserAdmin:** Configure the user administration
Further information: "Configuring the user administration", Page 293
- **OEM Function Users:** Edit the OEM function users
Further information: "HEIDENHAIN function users", Page 304
- **SELinux:** Define safety software for Linux-based operating systems
- **Shares:** Connect and manage external network drives
- **State Reporting Interface** (option 137): Activate the **SRI** and clear the status data
Further information: "State Reporting Interface (option 137)", Page 264
- **VNC:** Define the setting for external software accessing the control for e.g. maintenance work (**V**irtual **N**etwork **C**omputing)
Further information: "VNC", Page 267
- **WindowManagerConfig:** Available only to authorized specialists
- **Firewall:** Configure the firewall
Further information: "Firewall", Page 273
- **HePacketManager:** Available only to authorized specialists
- **HePacketManager Custom:** Available only to authorized specialists
- **Tools:** File applications
 - **Document Viewer:** Display and print files, e.g. PDF files
 - **File Manager:** Available only to authorized specialists
 - **Geeqie:** Open, manage, and print graphics
 - **Gnumeric:** Open, edit, and print tables
 - **Keypad:** Open virtual keyboard
 - **Leafpad:** Open and edit text files
 - **NC/PLC Backup:** Create backup file
Further information: "Backup and restore", Page 270
 - **NC/PLC Restore:** Restore backup file
Further information: "Backup and restore", Page 270
 - **QupZilla:** Alternative web browser for touch operation

- **Ristretto:** Open graphics
 - **Screenshot:** Create screenshots
 - **TNCguide:** Call up help system
 - **Xarchiver:** Extract or compress directories
 - **Applications:** Supplementary applications
 - **Orage Calender:** Open calendar
 - **Real VNC viewer:** Define the setting for external software accessing the control for e.g. maintenance work (Virtual Network Computing)
 - **Shutdown:** Shut down the control
- Further information:** "Switching or logging out users",
Page 315



The applications available under tools can be started directly by selecting the corresponding file type in the file management of the control

Further information: "Additional tools for management of external file types", Page 73

Portscan

The PortScan function enables the cyclic or manual searching for all open, incoming TCP and UDP list 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 **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 with machine manufacturer-specific functions, e.g. for Python and DNC 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.

Manually starting Portscan

Proceed as follows to manually start the Portscan:

- ▶ Taskbar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **Portscan** menu item
- > The control opens the **HeRos Portscan** pop-up window.
- ▶ Press the **Start** key

Cyclically starting Portscan

Proceed as follows to automatically start the Portscan cyclically:

- ▶ Taskbar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH 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** key
- ▶ Set the time interval with the slider

Remote Service

Together with the Remote Service Setup Tool, the TeleService from HEIDENHAIN enables encrypted end-to-end connections to be established between a service computer and the machine tool.

To enable the HEIDENHAIN control to communicate with the HEIDENHAIN server, the control needs to be connected to the Internet.

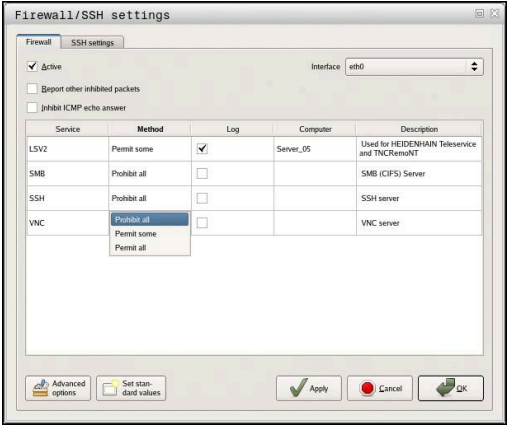
Further information: "General network settings", Page 283


In its basic state the firewall of the control blocks all incoming and outgoing connections. Therefore, the firewall settings must be adjusted or the firewall must be deactivated for the duration of the service session.

Setting up the control


Proceed as follows to deactivate the firewall:

- ▶ Taskbar at the bottom edge of the screen
- ▶ **Further information:** "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Firewall** menu item
- ▶ The control displays the **Firewall settings** dialog.
- ▶ Deactivate the firewall by removing the **Active** option in the **Firewall** tab.
- ▶ Press the **Apply** button to save the settings
- ▶ Press the **OK** button
- ▶ The firewall is disabled.





Do not forget to activate the firewall again after the end of the service session.



Alternative procedure for deactivating the firewall

The remote diagnosis via the TeleService PC software uses the **LSV2** service and this service must therefore be permitted in the firewall settings.

The following deviations from the firewall default settings are required:

- ▶ Set the method to **Permit some** for the **LSV2** service
- ▶ Enter the name of the service computer in the **Computer** column

The access security is ensured through the network settings. The network security is the responsibility of the machine tool builder or the respective network administrator.

Automatic installation of a session certificate

With an NC software installation a temporary certificate is automatically installed on the control. An installation, also in the form of an update, may only be carried out by a service technician from the machine tool builder.

Manual installation of a session certificate

A new certificate must be installed if no valid session certificate is installed on the control. Clarify which certificate is needed with your service employee. He will then provide you with a valid certificate file if necessary.

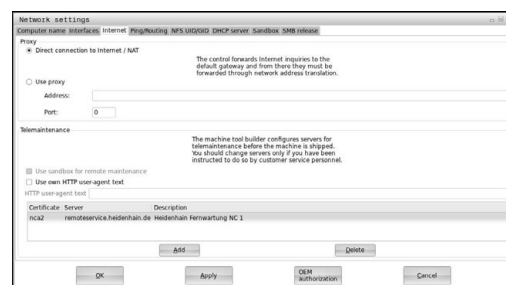
Proceed as follows to install the certificate on the control:

- ▶ Taskbar at the bottom edge of the screen
- ▶ **Further information:** "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Network** menu item
- ▶ The control displays the **Network settings** dialog.
- ▶ Select the **Internet** tab. The settings in the **Remote maintenance** field are configured by the machine tool builder.
- ▶ Press the **Add** button
- ▶ Select the file in the selection menu
- ▶ Press the **Open** key
- ▶ The certificate is opened.
- ▶ Press the **OK** soft key
- ▶ It may be necessary to restart the control to load the settings

Launching the service session

Proceed as follows to start the service session:

- ▶ Open the task bar at the bottom edge of the screen
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Diagnostic** menu item
- ▶ Select the **RemoteService** menu item
- ▶ Enter the **Session key** of the machine tool builder



Printer

The **Printer** function in the HeROS menu enables you to add and manage printers.

Displaying the printer settings

Proceed as follows to access the printer settings:

- ▶ Open the task bar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Printer** menu item
- The control opens the **Herros Printer Manager** pop-up window.

The name of the printer is displayed in the input field.

Soft key	Meaning
CREATE	Creates the printer named in the input field
CHANGE	Modifies the properties of the selected printer
COPY	Creates the printer named in the input field with the attributes of the selected printer This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP	Selects the desired printer
DOWN	
STATUS	Displays status information of the selected printer
PRINT	Prints a test page on the selected printer
TEST PAGE	

For each printer, the following attributes can be set:

Setting	Meaning
Name of the printer	The printer name can be changed here.
Connection	Here, you can select the connection type <ul style="list-style-type: none"> ■ USB: The USB connection can be assigned here. The name is displayed automatically. ■ Network: Enter the network name or the IP address of the target printer here. In addition, specify the port of the network printer here (default: 9100) ■ Printer not connected
Timeout	Defines the delay to printing after the last change has been made to the file to be printed in PRINTER:. This can be useful if the file to be printed is populated with data by using FN functions, e.g. during probing.
Standard printer	Select to define the standard printer in case several printers are available. Is defined automatically when creating the first printer.
Settings for printing of text	These settings are applicable when printing text documents: <ul style="list-style-type: none"> ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait, landscape for all printable files
Expert options	Available only to authorized specialists

Print options:

- Copying of the file to be printed in PRINTER:
The file to be printed is automatically forwarded to the standard printer and deleted from the directory after the print job has been executed
- Using the FN 16: F-PRINT function

List of printable files:

- Text files
- Graphic files
- PDF files



The connected printer must be PostScript-enabled.

SELinux security software

SELinux is an extension for Linux-based operating systems. SELinux is an additional security software package based on Mandatory Access Control (MAC) and protects the system against the running of unauthorized processes or functions and therefore protects 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 changed by programs that are 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 here to the relevant machine manual



HEIDENHAIN recommends activating SELinux because it provides additional protection against attacks from outside.

State Reporting Interface (option 137)

Introduction

With production lot sizes becoming ever smaller and products becoming increasingly customized, systems for production data acquisition are becoming increasingly important.

The operating resource data—as one of the most important parts of production data acquisition—describe the states of an operating resource along the time scale. For machine tools, the idle times and run times as well as information about pending faults are recorded. An evaluation per workpiece can also be made when the active NC programs are additionally taken into account.

One of the most common applications of production data acquisition is the determination of equipment effectiveness. The term Overall Equipment Effectiveness (OEE) is a measure of the value added of equipment. It provides information about the productivity of equipment as well as its losses at a glance.

With the **State Reporting Interface (SRI)**, HEIDENHAIN offers a simple and reliable interface for acquiring the operating states of your machine.

In contrast to other common interfaces, **SRI** also provides "historical" operating data. Even if your company network fails for several hours, your valuable operating data will not be lost.



A buffer memory that contains 2x 10 000 entries is available for storing the historical operating states. An entry corresponds to a status change in this context.

Configuring the control

Adjusting the firewall settings:

The **State Reporting Interface** uses **TCP Port 19090** to transfer the acquired operating states.

Accesses to the SRI from the company network (X26 connection) must be permitted in the firewall settings.

- Allow **SRI**

Further information: "Firewall", Page 273

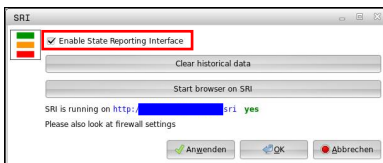


If the **SRI** is accessed locally via an IPC connected to the machine network (X116), eth0 (X26) can also remain locked.

Activating the **State Reporting Interface**:

In the control's factory default setting, the **SRI** is deactivated.

- Press the **DIADUR** key to open the HEROS menu
- Select the **Settings** menu item
- Select the **State Reporting Interface** menu item
- Activate the **State Reporting Interface** in the **SRI** pop-up window



Further information: "Overview of taskbar", Page 255



To clear all previous operating states, use the **Clear historical data** button.

Acquiring operating states

The **State Reporting Interface** uses the **Hypertext Transfer Protocol (HTTP)** to transfer the operating states.

With the following **URLs (Uniform Resource Locator)**, you can access the operating states of your control using any web browser.

- **http://<hostname>:19090/sri** for accessing all pieces of information (max. 20 000 entries)
- **http://<hostname>:19090/sri?lineno=<line>** for accessing the most recent pieces of information

Configuring **URL**:

- ▶ Replace the **<hostname>** by your control's network name
- ▶ Replace **<line>** by the first line to be retrieved
- ▶ The control transmits the requested data.

```
<html>
  <head></head>
  <body>
    <pre style="word-wrap: break-word; white-space: pre-wrap;">
      State Reporting Interface: 1.0.6
      HOST:      XXX
      HARDWARE: MC64XX 0.1
      SOFTWARE:  340590 09
      1 ; 2018-07-04 ; 09:52:22 ; TNC:\nc_prog\TS.h ; SUSPEND
      2 ; 2018-07-04 ; 09:52:28 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
      3 ; 2018-07-04 ; 09:52:30 ; TNC:\nc_prog\demo\Start_demo.h ; OPERATE
      4 ; 2018-07-04 ; 09:52:35 ; TNC:\nc_prog\demo\Start_demo.h ; ALARM
      5 ; 2018-07-04 ; 09:52:40 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
      6 ; 2018-07-04 ; 09:52:49 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
      7 ; 2018-07-04 ; 09:53:14 ; TNC:\nc_prog\demo\Start_demo.h ; SUSPEND
      8 ; 2018-07-04 ; 09:53:19 ; TNC:\nc_prog\demo\Start_demo.h ; OPERATE
      9 ; 2018-07-04 ; 09:53:24 ; TNC:\nc_prog\demo\Start_demo.h ; ALARM
    </pre>
  </body>
</html>
```

You will find the operating states in the **<body>** of the HTML file as **CSV** contents (**Comma Separated Values**).

CSV contents:

- Header

Designation	Meaning
State Reporting Interface:	Version of the interface. The version number should be taken into account in the evaluation of the data to ensure downward compatibility in your application.
SOFTWARE:	Software of the connected control.
HOST:	Full network name of the connected control.
HARDWARE:	Hardware of the connected control.

■ Operating data


Contents	Meaning
1	Consecutive number
2	
...	
2018-07-04	Date (yyyy-mm-dd)
09:52:22	Time (hh:mm:ss)
TNC:\nc_prog\TS.h	Selected or active NC program
States	Status:
■ OPERATE	■ Program run is active
■ SUSPEND	■ Program run stopped without error
■ ALARM	■ Program stopped due to an error

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:

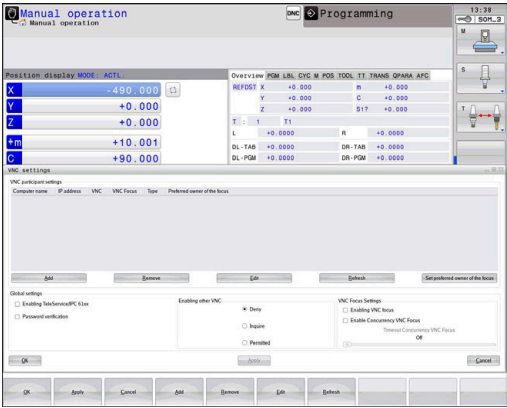
- List of permitted clients (IP address or name)
- Password for the connection
- Additional server options
- Additional settings for assigning the focus



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

Proceed as follows to open the VNC settings:




- ▶ Open the task bar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH 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 client 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	<ul style="list-style-type: none"> ■ 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		<p>Warnings and information about if the VNC protocol has not been authorized for all VNC clients due to firewall settings on the control.</p> <p>Further information: "Firewall", Page 273.</p>
Global settings	Enable TeleService 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.
	Enabling 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 makes it possible to refuse that the focus be given to the requesting client. If this does not occur, the focus changes to the requesting client after the set time limit.

Backup and restore

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the complete **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 a ***. tncbck** file is selected in the file manager of the control, the program **NC/PLC Restore** is automatically launched.

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.

Opening NC/PLC Backup or NC/PLC Restore

Proceed as follows to open the functions:

- ▶ Open the task bar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH 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 backup data from the control, proceed as follows:

- ▶ Select **NC/PLC Backup**
- ▶ Select the type
 - Back up the **TNC** partition
 - Back up the directory tree: Select the directory for backup in the file management
 - Back up the machine configuration (for machine tool builders only)
 - Complete backup (for machine tool builders 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
 - Use 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.

Restoring data

NOTICE

Caution: Data may be lost!

When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted.

- ▶ Before starting the data restore process, make a backup of the existing data

To restore the data proceed as follows:




- ▶ 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 can set up a firewall for the primary network interface of the control. It can be configured 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.

Once the firewall has been activated, a symbol appears at the lower 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
	Firewall protection does not yet exist although it has been activated according to the configuration. This can happen, for example, if PC names for which there are no equivalent IP addresses as yet were used in the configuration.
	Firewall active with medium security level
	Firewall active with high safety level. (All services except for the SSH are blocked)



Have your network specialist check and, if necessary, change the standard settings.

Configuring the firewall


Make your firewall settings as follows:

- ▶ Use the mouse to open the task bar at the bottom edge of the screen
Further information: "Window manager", Page 254
- ▶ Press the green HEIDENHAIN button to open the JH menu
- ▶ Select the **Settings** menu item
- ▶ Select the **Firewall** menu item.

HEIDENHAIN recommends activating the firewall with the prepared default settings:

- ▶ Select the **Active** option to enable the firewall
- ▶ Press the **Set standard values** button to activate the default settings recommended by HEIDENHAIN.
- ▶ Confirm the changes with **Apply**
- ▶ Close the dialog with **OK**.

Firewall settings

Option	Meaning
Active	Switching the firewall on and off
Interface	<p>Selection of the eth0 interface usually corresponds to X26 of the MC main computer. eth1 corresponds to X116. You can check this 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.</p> <div>  <p>To configure the sandbox, you use the optional interface brsb0. Further information: "Sandbox tab", Page 288</p> </div>
Report other inhibited packets	Firewall active with high safety level. (All services except for the SSH are blocked)
Inhibit ICMP echo answer	If this option is set, the control no longer responds to a PING request
Service	<p>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</p> <ul style="list-style-type: none"> ■ LDAPS includes the server on which the user data and the user administration configuration are saved. ■ LSV2 includes the functionality for TNCremo and Teleservice, as well as the HEIDENHAIN DNC interface (ports 19000 to 19010) ■ SMB only refers to incoming SMB connections, i.e. if a Windows release is made on the NC. Outgoing SMB connections (i.e. if a Windows release is connected to the NC) cannot be prevented. ■ SRI refers to the connections that are used in conjunction with the acquisition of operating states through the State Reporting Interface option. ■ 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: "DNC connection with user authentication", Page 310 ■ VNC protocol means access to the screen contents. If this service is blocked, the screen content can no longer be accessed, not even with the TeleService programs from HEIDENHAIN (e.g. screenshot). If this service is blocked, the VNC configuration dialog shows a warning from HEROS that VNC is disabled in the firewall.
Method	Use Method to configure whether the service is inaccessible for all (Prohibit all), accessible for all (Permit all) or only accessible for individuals (Permit some). If you set Permit some you must also specify the computer (under Computer) that you wish to grant access to the respective service. If you do not specify any computer under Computer , the setting Prohibit all will automatically become active when the configuration is saved.
Log	If Log is activated, a red message is output if a network packet for this service was blocked. A (blue) message is output if a network packet for this service was accepted

Option	Meaning
Calculator	If the setting Permit some is selected under Method , the relevant computers can be specified here. The computers can be entered with their IP addresses or host names separated by commas. If a host name is used, the system checks upon closing or saving of the dialog whether the host name can be translated into an IP address. If this is not the case, an error message is displayed and the dialog does not terminate. If a valid host name is specified, 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 to ensure that 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	Resets the settings to the default values recommended by HEIDENHAIN

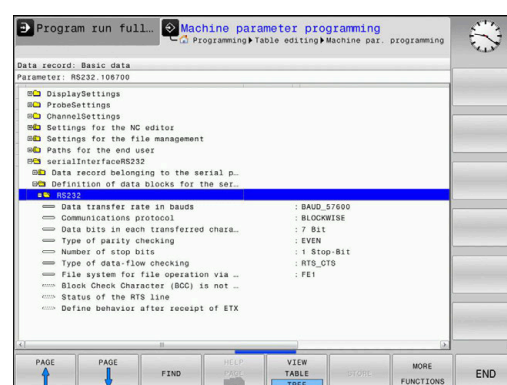
9.3 Setting up data interfaces

Serial interfaces on the TNC 128

The TNC 128 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is prescribed and cannot be modified apart from setting the baud rate (machine parameter **baudRateLsv2**, no. 106606). You can also define another type of data transfer (interface). The settings described below are therefore effective only for the respective newly defined interface.

Application

To set up a data interface, press the **MOD** key. Enter the code number 123. In the **CfgSerialInterface** (no. 106700) machine parameter, you can enter the following settings:



Setting the RS-232 interface

Open the RS232 folder. The control then displays the following settings:

Set BAUD RATE (baud rate no. 106701)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Set protocol (protocol no. 106702)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).



Operating notes:

- 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 nor to the simultaneous execution of older contouring controls in blocks. This function is no longer available for current controls.

Data transmission protocol	Selection
Standard data transmission (transmission line-by-line)	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol (only character-by-character)	RAW_DATA

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 helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

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.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active through RTS
- Software handshaking (XON_XOFF): Transmission stop is active through DC3 (XOFF)

File system for file operation (fileSystem no. 106707)

In **fileSystem** you define the file system for the serial interface. This machine parameter is not required if you don't need a special file system.

- EXT: Minimum file system for printers or non-HEIDENHAIN 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)

With Block Check Character (optional) no control character, you determine whether the checksum can correspond to a control character.

- TRUE: The checksum does not correspond to a control character
- FALSE: The checksum can correspond to a control character

Condition of RTS line (rtsLow no. 106709)

With the state of the RTS line (optional), you can define whether the **LOW** level is active in idle state.

- TRUE: Level is **LOW** in idle state
- FALSE: Level is not **LOW** in idle state

Define behavior after receipt of ETX (noEotAfterEtx no. 106710)

With define behavior after reception of ETX (optional) you determine whether the EOT character is sent after the ETX character was received.

- TRUE: The EOT character is not sent
- FALSE: The EOT character is sent

Settings for the transmission of data using PC software TNCserver

Apply the following settings in machine parameter **RS232** (no. 106700):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external device (fileSystem)



The **load all programs**, **load offered program**, and **load directory** functions are not available in the **FE2** and **FEX** operating modes.

Icon	External device	Operating mode
	PC with TNCremo software	LSV2
	HEIDENHAIN floppy disk units	FE1
	Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without TNCremo	FEX

Software for data transfer

For data transfer to or from the control, you should use the **TNCremo** software. With **TNCremo**, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.



You can download the current version of the **TNCremo** software from the HEIDENHAIN homepage.

System requirements for TNCremo:

- PC with 486 processor or higher
- Windows XP, Windows Vista, Windows 7, Windows 8 operating system
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- ▶ Start the SETUP.EXE installation program with the file manager (Explorer)
- ▶ Follow the setup program instructions

Starting TNCremo under Windows

- ▶ Click <Start>, <Programs>, <HEIDENHAIN Applications>, <**TNCremo**>

When you start **TNCremo** for the first time, **TNCremo** automatically tries to set up a connection with the control.

Data transfer between the control and TNCremo

Check whether the control is connected to the correct serial port on your PC or to the network.

Once you have started the **TNCremo** software, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using <File>, <Change directory>, you can select any drive or another directory on your PC.

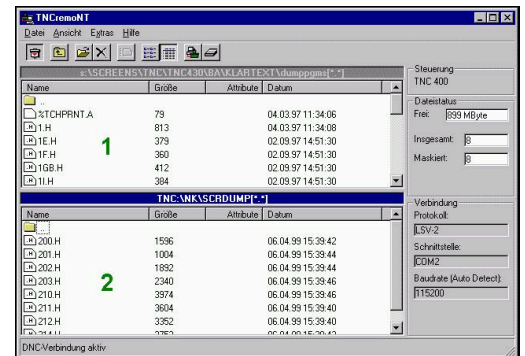
If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- ▶ Select <File>, <Setup connection>. **TNCremo** now receives the file and directory structure from the control and displays this in the lower part of the main window **2**
- ▶ To transfer a file from the control to the PC, select the file in the control window per mouse click and move the highlighted file into the PC window while holding down the mouse button **1**
- ▶ To transfer a file from the PC to the control, select the file in the PC window per mouse click and move the highlighted file into the control window while holding down the mouse button **2**

If you want to control data transfer from the control, establish the connection with your PC in the following manner:

- ▶ Select <Extras>, <TNCserver>. **TNCremo** then starts in server mode and can receive data from the control or send data to the control
- ▶ You can now call the file management functions on the control by pressing the **PGM MGT** key in order to transfer the desired files

Further information: "Data transfer to or from an external data carrier", Page 69



If you have exported a tool table from the control, then the tool types are converted to tool type numbers.

End TNCremo

Select <File>, <Exit>



You can open the context-sensitive help function of the **TNCremo** software by pressing the **F1** key.

9.4 Ethernet interface

Introduction

The control is provided with an Ethernet card as a standard feature so that you can integrate it into your network as a client.

The control transmits data via the Ethernet card using the following protocols:

- the **smb** protocol (Server Message Block) for Windows operating systems, or
- The **TCP/IP** protocol family (**T**ransmission **C**ontrol **P**rotocol/**I**nternet **P**rotocol) and with support from the NFS (**N**etwork **F**ile **S**ystem)



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

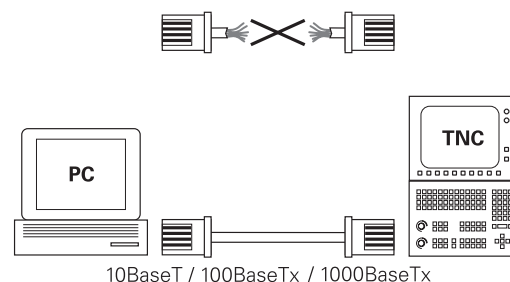
Connection possibilities

You can connect the Ethernet card in your control to your network through the RJ45 connection (X26, 1000BaseTX, 100BaseTX and 10BaseT), or directly to a PC. The connection is metalically isolated from the control electronics.


For a 1000Base TX, 100BaseTX, and 10BaseT connection, use a twisted-pair cable to connect the control to your network.



The maximum possible cable length depends on the quality grade of the cable, the sheathing, and the type of network (1000BaseTX, 100BaseTX, or 10BaseT)



General network settings

 Have a network specialist configure the control.

Proceed as follows to access the general network settings:

- MOD


▶ Press the **MOD** key
- PGM MGT

▶ Enter the code number **NET123**
▶ Press the **PGM MGT** key
- NET

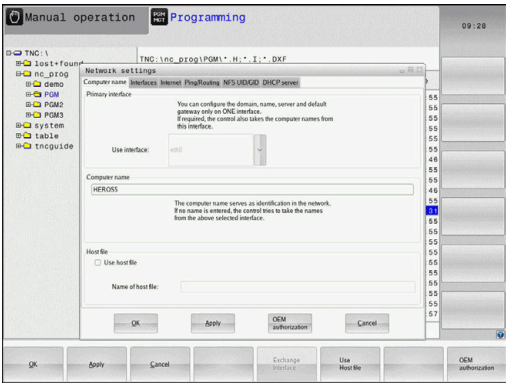
▶ Press the **NET** soft key
- CONFIGURE NETWORK

▶ Press the **CONFIGURE NETWORK** soft key

Computer name tab

 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 language.

Setting	Meaning
Primary interface	Name of the Ethernet interface to be integrated in your company network. Only active if a second, optional Ethernet interface is available on the control hardware
Computer name	Name displayed for the control in your company network
Host file	Only required for special applications: Name of a file in which the assignments of IP addresses to computer names are defined



Interfaces tab

Setting	Meaning
Interface list	<p>List of the active Ethernet interfaces. Select one of the listed interfaces (via mouse or arrow keys)</p> <ul style="list-style-type: none"> ■ Activate button: Activate the selected interface (X appears in the Active column) ■ Deactivate button: Deactivate the selected interface (- appears in the Active column) ■ Configuration button: Open the configuration menu

Allow IP forwarding	<p>This function must be kept deactivated. Only activate this function after instruction by our Service Department for diagnostic purposes. If external access to the second, optional Ethernet interface is necessary, this function needs to be activated.</p>
----------------------------	---

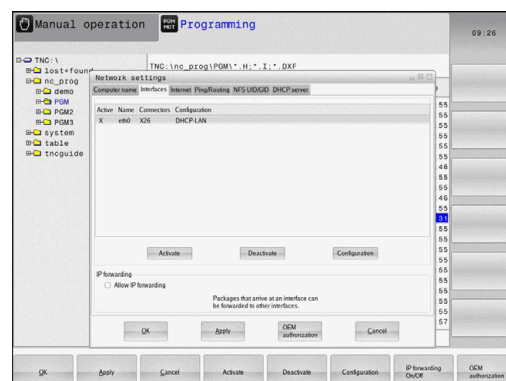
Proceed as follows to access the configuration menu:

- Press the **Configuration** button

Setting	Meaning
Status	<ul style="list-style-type: none"> ■ Interface active: Connection status of the selected Ethernet interface ■ Name: Name of the interface you are currently configuring ■ Plug connection: Number of the plug connection of this interface on the logic unit of the control

Profile	<p>Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles:</p> <ul style="list-style-type: none"> ■ DHCP-LAN: Settings for the standard Ethernet interface; should work in a standard company network ■ MachineNet: Settings for the second, optional Ethernet interface; for configuration of the machine network <p>Press the corresponding buttons to save, load and delete profiles</p>
----------------	---

IP address	<ul style="list-style-type: none"> ■ Automatically procure IP address: The control is to procure the IP address from the DHCP server ■ Manually set IP address: Manually define the IP address and subnet mask. Input: Four numerical values separated by periods, e.g. 160.1.180.20 and 255.255.0.0
-------------------	--

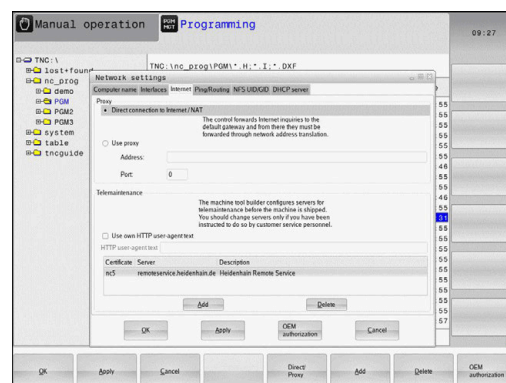


Setting	Meaning
Domain Name Server (DNS)	<ul style="list-style-type: none"> ■ Automatically procure DNS: The control is to automatically procure the IP address of the domain name server ■ Manually configure the DNS: Manually enter the IP addresses of the servers and the domain name
Default gateway	<ul style="list-style-type: none"> ■ Automatically procure default gateway: The control is to automatically procure the default gateway ■ Manually configure the default gateway: Manually enter the IP addresses of the default gateway

- Apply the changes with the **OK** button, or discard them with the **Cancel** button

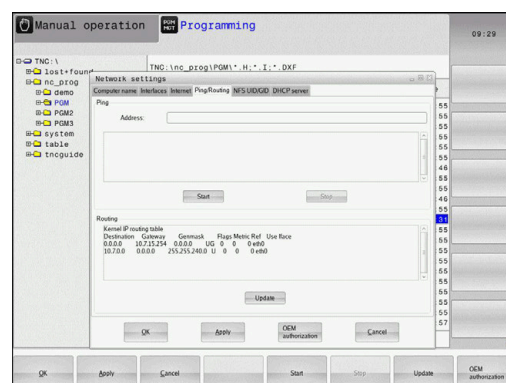
Internet tab

The **Internet** tab currently has no function.



Ping/Routing tab

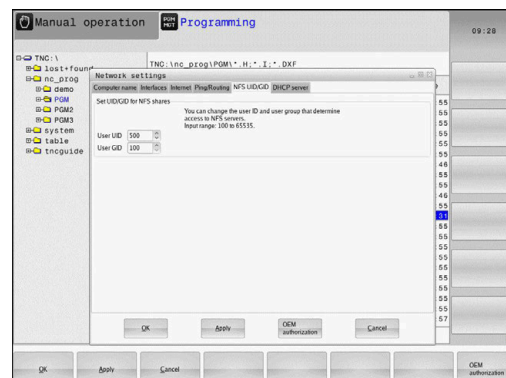
Setting	Meaning
Ping	<p>In the Address: input field, enter the IP number for which you want to check the network connection. Input: Four numerical values separated by periods, e.g. 160.1.180.20. As an alternative, you can enter the name of the computer whose connection you want to check</p> <ul style="list-style-type: none"> ■ Press the Start button to begin the test. The control shows the status information in the Ping field ■ Press the Stop button to conclude the test
Routing	<p>For network specialists: Status information of the operating system for the current routing</p> <ul style="list-style-type: none"> ■ Press the Update button to refresh the routing information



NFS UID/GID tab

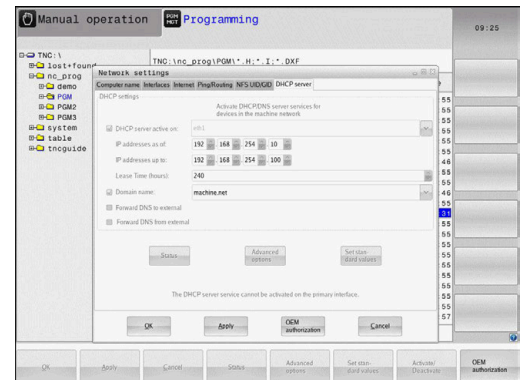
On the **NFS UID/GID** tab, you enter the user and group identifications.

Setting	Meaning
Set UID/GID for NFS shares	<ul style="list-style-type: none"> ■ User ID: Definition of which user identification the end user uses to access files in the network. Ask your network specialist for the proper value ■ Group ID: Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value



DHCP Server tab

Setting	Meaning
DHCP server	<ul style="list-style-type: none"> ■ IP addresses from: Define the IP address as of which the control is to derive the pool of dynamic IP addresses. The control transfers the values that appear dimmed from the static IP address of the defined Ethernet interface; these values cannot be edited. ■ IP addresses to: Define the IP address up to which the control is to derive the pool of dynamic IP addresses. ■ Lease Time (hours): Time within which the dynamic IP address is to remain reserved for a client. If a client logs on within this time, the control reassigns the same dynamic IP address. ■ Domain name: Here you can define a name for the machine network if required. This is necessary if the same names are assigned in the machine network and in the external network, for example. ■ Forward DNS to external: If IP Forwarding is active (Interfaces tab) and the option is active, you can specify that the name resolution for devices in the machine network can also be used by the external network. ■ Forward DNS from external: If IP Forwarding is active (Interfaces tab) and the option is active, you can specify that the control is to forward DNS inquiries from devices within the machine network to the name server of the external network if the DNS server of the MC cannot answer the inquiry. ■ Status button: Call an overview of the devices that are provided with a dynamic IP address in the machine network. You can also select settings for these devices. ■ Advanced options button: Additional settings for the DNS/DHCP server. ■ Set standard values button: Set factory default settings.



Sandbox tab

On the **Sandbox** tab, you configure the sandbox feature.

With the sandbox feature, the control enables you to execute applications in an environment that is isolated from the rest of the control. By isolating the data accesses, applications that run within a sandbox container have no access to files outside the virtual environment. This can be used, for example, to run a browser with access to the Internet.



Configure and use the sandbox on your control. For safety and security reasons, always open the browser in the sandbox.

Proceed as follows to activate the sandbox:

- ▶ Activate the Sandbox option (by selecting it)
- > The control activates the default settings for the sandbox.
- > When the default settings are active, the control offers you to start the browser from within the sandbox.

The sandbox can share a network connection (e.g. eth0) with the control. You can use the **Configure** button to configure separate network settings for the sandbox.



You can define firewall settings for the sandbox using the **brsb0** interface.

Further information: "Firewall", Page 273

With the aid of the network settings, this enables you to allow only the sandbox to access the Internet. The control is only allowed to access its local Intranet or machine network. In this case, the browser is only allowed to access the Internet if the browser runs within the sandbox.

The sandbox is automatically given its own computer name. The computer name of the control is therefore suffixed with **_sandbox**.

Device-specific network settings



Have a network specialist configure the control.

You can define any number of network settings, but you can manage only seven at one time.

Proceed as follows to access the device-specific network settings:

MOD

- ▶ Press the **MOD** key

Alternative:

PGM
MGT

- ▶ Press the **PGM MGT** key

NET

- ▶ Press the **NET** soft key

DEFINE
NETWORK
CONNECTN.

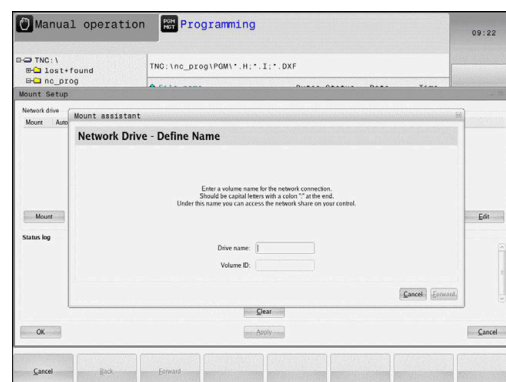
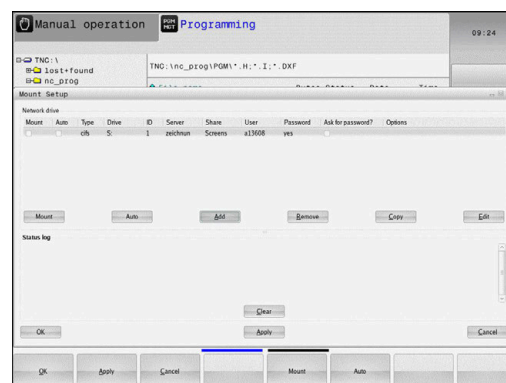
- ▶ Press the **DEFINE NETWORK CONNECTN.** soft key

To manage the network drives, use the screen buttons.

Proceed as follows to add a network drive:

- ▶ Press the **Add** button
- > The control starts the connection wizard, which guides you by dialog through the required definitions.

Setting	Meaning
Network drive	<p>List of all connected network drives. The control shows the respective status of the network connections in the columns:</p> <ul style="list-style-type: none"> ■ Mount: Network drive connected / not connected ■ Auto: Network drive is to be connected automatically/manually ■ Type: Type of network connection. cifs and nfs are possible ■ Drive: Designation of the drive on the control ■ ID: Internal ID that identifies if a mount point has been used for more than one connection ■ Server: Name of the server ■ Share: Name of the directory on the server that the control is to access ■ User: User name with which the user logs on to the network ■ Password: Network password protected or not ■ Query password?: Query / do not query password during connection ■ Options: Display additional connection options
Status log	<p>Display of status information and error messages.</p> <p>Press the Clear button to delete the contents of the Status Log window.</p>



9.5 SELinux security software

SELinux is an extension for Linux-based operating systems. SELinux is an additional security software package based on Mandatory Access Control (MAC) and protects the system against the running of unauthorized processes or functions and therefore protects 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 changed by programs that are 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 here to the relevant machine manual



HEIDENHAIN recommends activating SELinux because it provides additional protection against attacks from outside.

9.6 User administration



Refer to your machine manual.

Some user administration areas are configured by the machine tool builder.

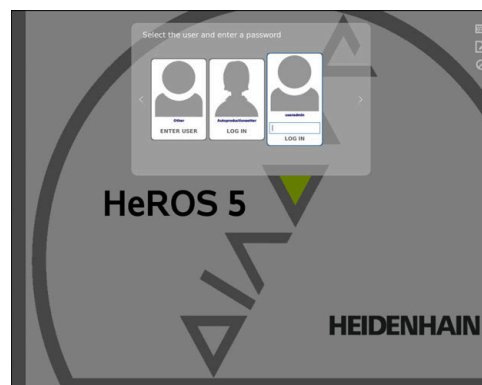
If you want to use the user administration feature on a control without HEIDENHAIN keyboard, you need to connect an external alphabetic keyboard to the control.

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 the user administration feature, 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 the user administration feature 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 295

- **LDAP database on remote computer**

- Importing an LDAP server configuration file

Further information: "LDAP on remote computer", Page 296

- **Connection to a Windows domain**

- Integrating the user administration feature on more than one control
 - Using different roles on different controls

Further information: "Connection to a Windows domain", Page 297



Parallel operation of Windows users and users from an LDAP database is possible.

Configuring the user administration

User administration is inactive in the control's factory default setting. This status is called **legacy mode**.

Further information: "", Page 292

Before user administration can be used, you need to configure it. Perform the following steps for configuration:

1. Activate the user administration and create the **useradmin** user
2. Set up a database

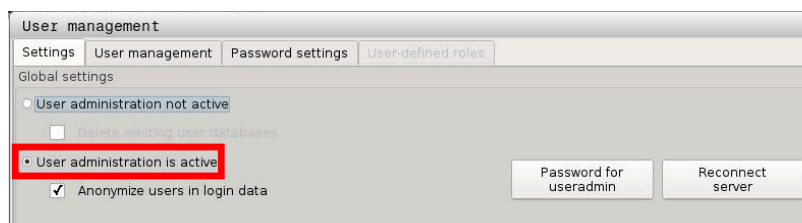
- **Further information:** "Local LDAP database", Page 295
- **Further information:** "LDAP on remote computer", Page 296
- **Further information:** "Connection to a Windows domain", Page 297

3. Create further users

Further information: "Creating further users", Page 300

Proceed as follows to call the user administration:

- ▶ Select the **HEROS** menu icon
- ▶ Select the **Settings** menu item
- ▶ Select the **UserAdmin** menu item
- > The control opens the **User management** window.
- 1. Proceed as follows to activate the user administration:
 - ▶ Call the user administration
 - ▶ Press the **User administration active** soft key
 - > The control shows the message **Password for user 'useradmin' missing**.



The function **Anonymize users in login data** is active in the factory default setting. As a consequence, the user name is blacked out in all log data. If your company permits the filing of personal data on IT systems, you can deactivate this function.

After user administration has been activated, create the **useradmin** function user.

Proceed as follows to create the **useradmin** user:

- ▶ Press **Password for useradmin**
- > The control opens the **Password for user useradmin** pop-up window.
- ▶ Enter the password for the **useradmin** user
- ▶ Select **Set new password**
- > The message **Settings and password for useradmin were changed** appears.



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

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating the user administration



The **useradmin** user automatically receives the **HEROS.Admin** role. This allows him to manage users in the user administration, provided that he knows the LDAP database password. The **useradmin** user is a function user who has been predefined by HEIDENHAIN. You can neither add roles to function users, nor delete their roles.

Further information: "Definition of roles", Page 305
HEIDENHAIN recommends permitting more than one person to access an account with the **HEROS.Admin** role. This ensures that necessary changes to the user administration can also be made in the administrator's absence.

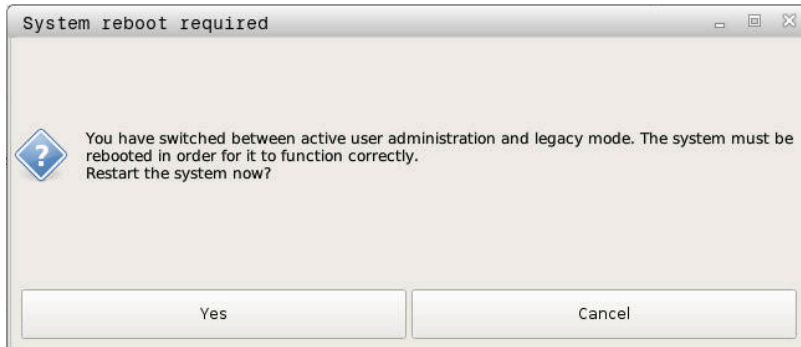
2. Proceed as follows to set up the database:

- ▶ Select the database for saving the user data
 - Local LDAP database

Further information: "Local LDAP database", Page 295
 - LDAP on remote computer

Further information: "LDAP on remote computer", Page 296
 - Connection to a Windows domain

Further information: "Connection to a Windows domain", Page 297
- ▶ 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 is restarted.



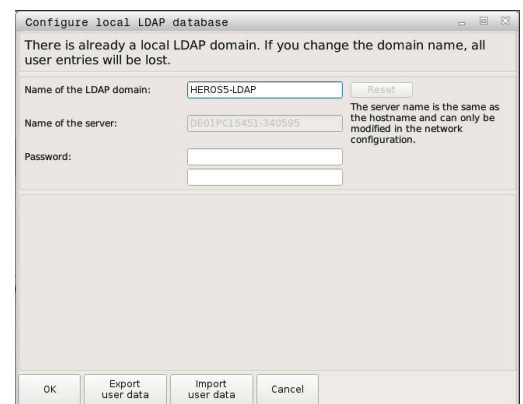
Local LDAP database

Before you can use the **Local LDAP database** function, the following requirements must be met:

- The user administration is active
- The **useradmin** user has already been configured

Proceed as follows 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 **Configure** 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 300

LDAP on remote computer

Before you can use the **LDAP on remote computer** function, the following requirements must be met:

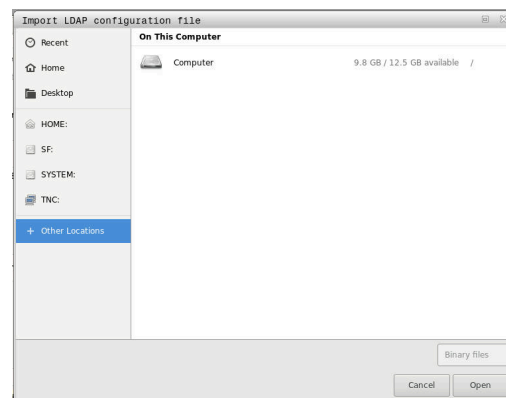
- The user administration is active
- The **useradmin** user has been configured
- An LDAP database has been set up in the company network
- A server configuration file of an existing LDAP database must be stored on the control or on a PC in the network
- The PC with the existing configuration file is running.
- The PC with the existing configuration file is accessible in the network

Proceed as follows to provide an LDAP database with a server configuration file:

- ▶ 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

Proceed as follows 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 **Open**
- ▶ Press the **APPLY** soft key.
- > The configuration file was imported.



Connection to a Windows domain

Before you can use the **Connection to Windows domain** function, the following requirements must be met:

- The user administration is active
- The **useradmin function user** has already been set up
- There is a Windows Active Domain Controller in the network
- 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
- The domain controller is accessible in the network

Proceed as follows to set up the **Connection to Windows domain** function:

- ▶ Call the user administration
- ▶ Select the **Connection to Windows domain** function
- ▶ Select the **Find domain** function
- > The control recognizes the found domain.



Use the **Configure** function to define various settings of your connection:

- Deactivate the function **Map SIDs to Unix UIDs**
- You can define a special group of Windows users to whom you want to restrict the connection to this control
- You can modify the organizational unit in which the HEROS role names are stored
- You can 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
- You can change the separator within the HEROS role name

- ▶ Press the **APPLY** soft key.
- > The control opens the **Connect to domain** window.



With the **Organizational unit for computer account** function, you can specify in which of the already existing organizational units you want to create the access, e.g.

- ou = controls
- cn = computers

The values you enter must match the conditions of the domain. The terms are not exchangeable.

- ▶ Enter the user name of the domain controller
- ▶ Enter the password of the domain controller
- > 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.

- Select the **Export** function

You can output the roles externally to a file in .ldif format in this case.

- > All of the required roles have been created in the domain as groups.

☐ Connection to Windows domain

Domain: KDC:
LDAP ID-mapping: Yes

HEROS role base:

Configuration Find domain Add role definition

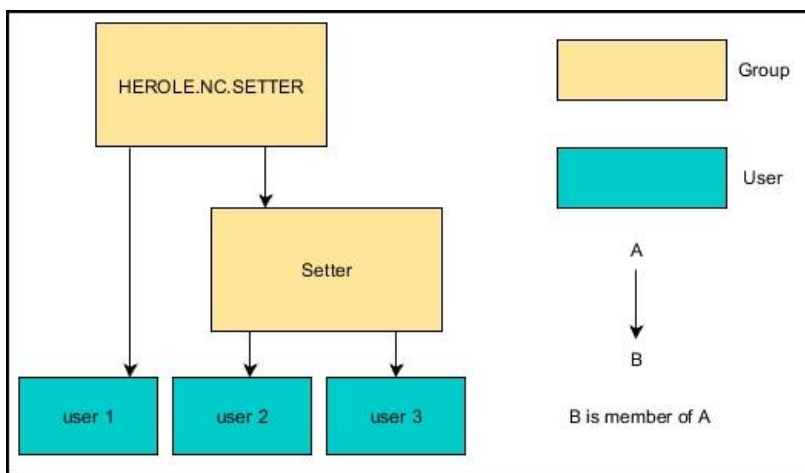
There are the following ways to create groups corresponding to the different roles:

- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format into the Windows server

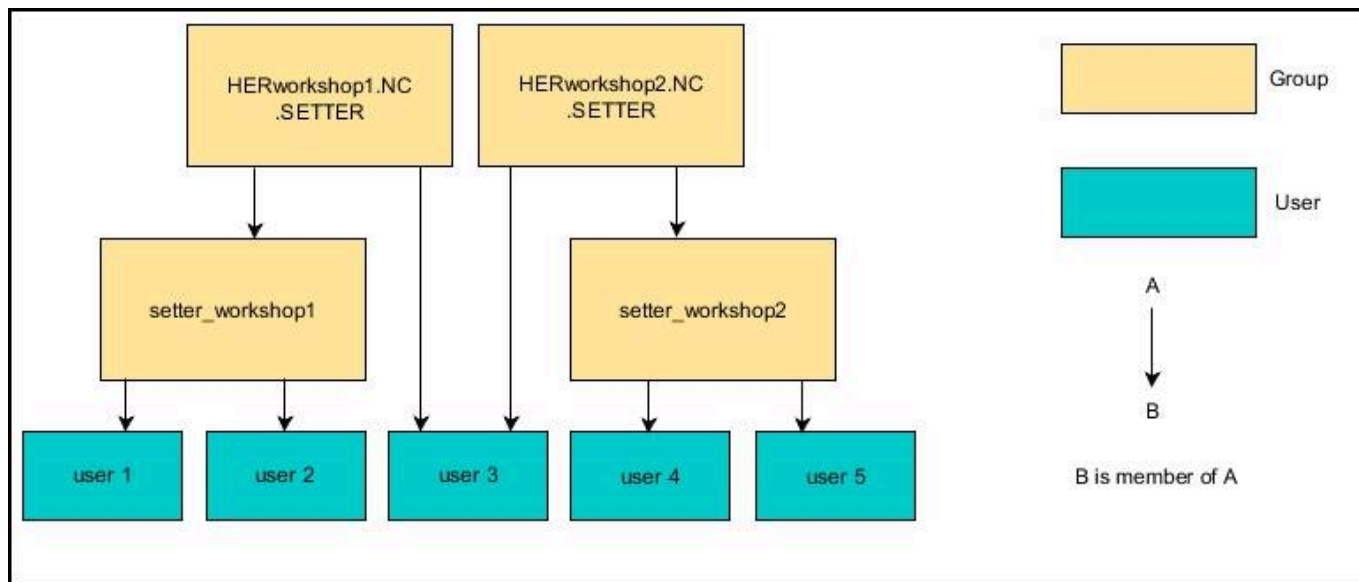
The users need to be added manually by the Windows administrator to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by HEIDENHAIN below:

- Suggestion 1: The user is a direct or indirect member of the respective group:



- Suggestion 2: Users from various sectors (workshops) are members of groups with different prefixes:



Creating further users

After configuring the user administration, you can create further users.

Before you can create further users, you must have configured and selected an LDAP database.

Proceed as follows to create further users:

- ▶ Call the user administration
- ▶ Select the **User management** tab



The **User management** tab is only effective 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: "Connection to a Windows domain", Page 297

- ▶ Press the **EDIT ON** soft key
- ▶ The control prompts you to enter the password of your user database.



This step is only needed if you have restarted your control after configuring the database.

- ▶ After you have entered the password, the control opens the **User management** menu.

You can edit the existing users or create new users.

Proceed as follows to create a new user:

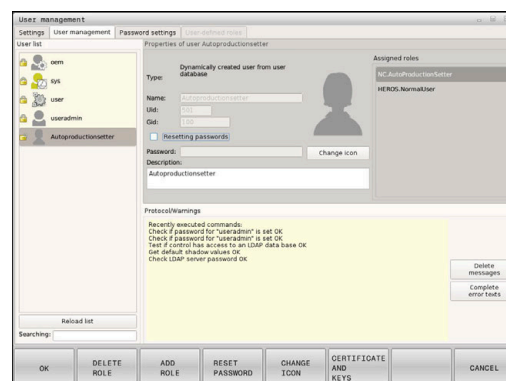
- ▶ Press the **CREATE NEW USER** soft key
- ▶ The control opens a window for creating a user.
- ▶ Enter the user name
- ▶ Enter a password for the user



The user has to change this password the first time he logs on.

Further information: "Login to user administration", Page 314

- ▶ You can optionally also create a user description.
- ▶ Press the **ADD ROLE** soft key
- ▶ In the selection window, select the roles corresponding to your user. **Further information:** "Definition of roles", Page 305
- ▶ Confirm your selection with the **ADD** soft key





Two additional soft keys are available in the menu:

■ **ADD REMOTE 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.

Further information: "Definition of roles", Page 305

- ▶ 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 changes you made are applied.
- ▶ Press the **END** soft key
- > The control closes the user administration.



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 the user administration", Page 293

You can optionally also assign images to your users. The **default user icons** from HEIDENHAIN are available for this. 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.

Proceed as follows to define profile images:

- ▶ Log on the user with **HEROS.Admin** role, e.g. **useradmin**

Further information: "Login to user administration", Page 314

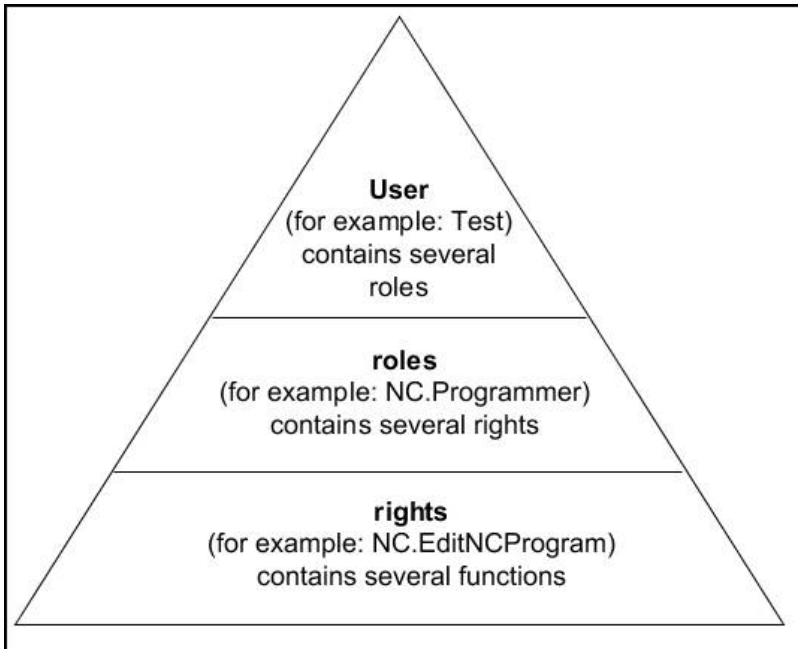
- ▶ Call the user administration
- ▶ Select the **User management** tab
- ▶ Press the **EDIT USER** soft key
- ▶ Press the **CHANGE ICON** soft key
- ▶ Select image
- ▶ Use the **CHOOSE ICON** soft key to select an image
- ▶ Press the **OK** soft key
- ▶ Press the **APPLY** soft key.
- > The changes you made are applied.



You can also directly add profile images when creating the users.

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

- **Function user** pre-defined by HEIDENHAIN

Further information: "HEIDENHAIN function users",
Page 304

- **Function user** defined by the machine tool builder
- Self-defined users

A user can be pre-defined in the control or he can be defined by the end user. A user contains all the roles that have been assigned to him.



Your machine tool builder defines function users who are required for machine maintenance, for example. 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.

Further information: "Creating further users",
Page 300

Roles consist of a combination of rights that cover certain functions provided by the control.

Roles:

- **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. If more than one role is assigned to a user, he will be granted all rights contained in these roles.

Rights:

- HEROS rights
- NC rights
- PLC rights (OEM)

Rights consist of a combination of functions that cover a task area of the control, e.g. editing the tool table.



Ensure that every user is only assigned the access rights he needs. The access rights result from the actions a user performs on and with the control.

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created when the user administration is activated. Function users cannot be edited.

HEIDENHAIN provides four different function users in the control's factory default setting.

■ **oem**

The **oem** function user is intended for the machine tool builder. With **oem**, you can access the PLC partition of the control.

■ **Function user defined by the machine tool builder**



Refer to your machine manual.

The users defined by the machine tool builder can differ from those pre-defined by HEIDENHAIN.

The machine tool builder'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 318



■ **sys**

With the **sys** function user, you can access the system partition of the control. This function user is reserved for use by the JH Service department.

■ **user**

In **legacy mode**, the **user** function user is automatically logged on to the system during control start-up. When the 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 when the user administration is activated. With **useradmin**, you can configure and edit the 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.



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 he uses 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**.

Further information: "Creating further users",
Page 300

Advantages of classification in roles:

- Easier administration for the end user
- Different rights are compatible between different software versions of the control and different machine manufacturers.



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**.



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	Rights		
	HEROS role name	UNIX group	Basic ID number
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 332
	■ 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	■ 331
HEROS.LegacyUser	■ HEROS.ControlFunctions	■ ctrlfct	■ 337
	With the legacy user role, the behavior regarding the operating system of the control is identical to that of older software versions without user administration. The user administration remains active.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 334
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.SWUpdate	■ swupdate	■ 338
	■ HEROS.SetNetwork	■ netadmin	■ 333
	■ HEROS.SetTimezone	■ tz	■ 330
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.Admin	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the LegacyUser role, as well as the following rights:		
	■ HEROS.UserAdmin	■ useradmin	■ 336

NC operator roles:

Role	Rights		
	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 legacy user role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. The 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.		
	■ FN 17 and editing the table head		
	Replacement for code number 555343		
	■ NC.EditNCProgramAdv	■ NCEditNCPgmAdv	■ 327
	■ NC.EditTableAdv	■ NCEditTableAdv	■ 328
NC.RemoteOperator	This role allows NC program start via the DNC interface.		
	■ NC.RemoteProgramRun	■ NCRemotePgmRun	■ 329

Machine tool builder (PLC) roles:

Role	Rights		
	HEROS role name	UNIX group	Basic ID number
PLC.ConfigureUser	This role grants the rights on code number 123 .		
	■ NC.ConfigUserAdv	■ NCConfigUserAdv	■ 316
	■ NC.SetupDrive	■ NCSetupDrv	■ 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 tool builder can adapt the PLC roles.

When the **machine tool builder (PLC) roles** are adapted by the machine tool builder, 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
NC.OPModeManual	Operation of 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	Program a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes
NC.EditNCProgramAdv	Additional NC functions, e.g. FN 17
NC.EditTableAdv	Additional table programming functions, e.g. editing the table head
HEROS.SetTimezone	Adjustment of time and date, time zone, and time synchronization via NTP and HEROS menu
HEROS.SetShares	Configuration of network shares mounted by 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 <ul style="list-style-type: none"> ■ 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

HEROS role name	Description
NC.RemoteProgramRun	Start of NC program via remote interface, e.g. DNC
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number 123
NC.Data.AccessServiceRead	Read-only access to the PLC partition during servicing

DNC connection with user authentication

Introduction

If user administration is active, DNC applications also need to authenticate a user for the correct rights to be assigned.

To do this, the connection is routed over 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.

Besides, the encryption used with the SSH tunnel protects the communication from attackers.

Concept of transmission through an SSH tunnel

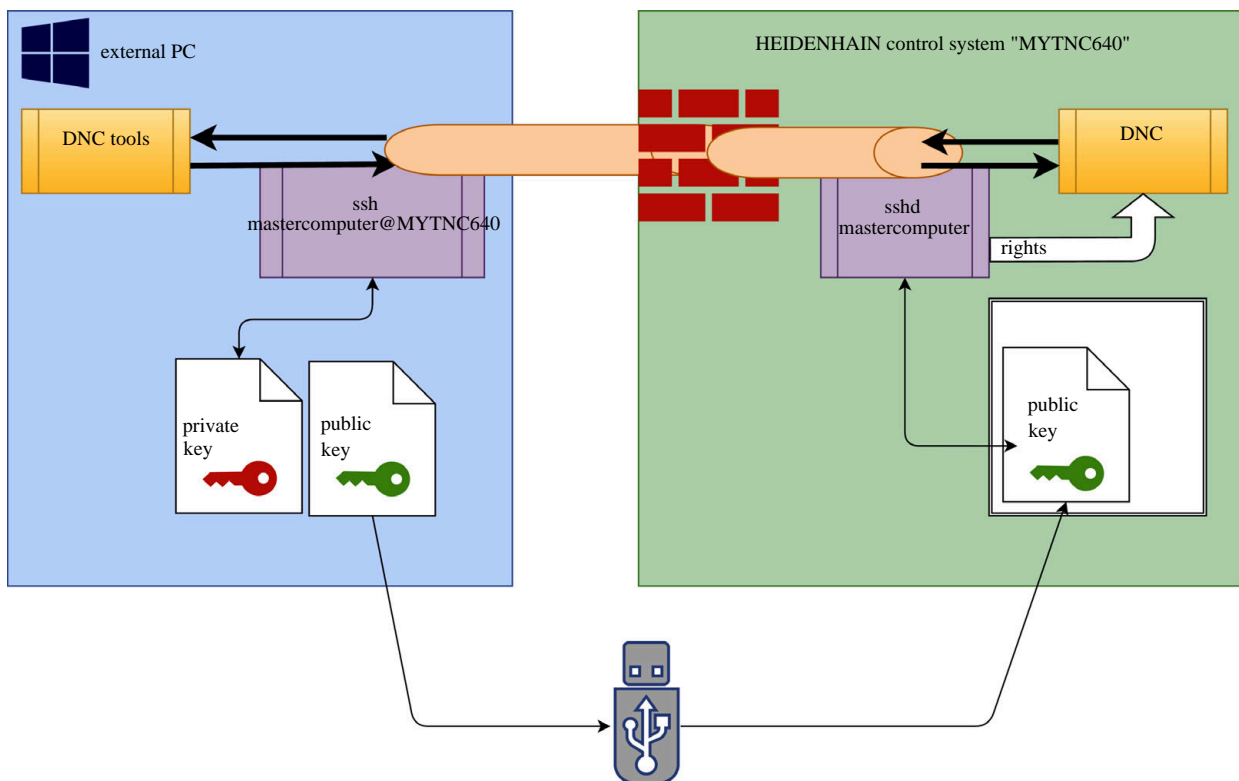
Prerequisites:

- 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 DNC tools

The PC tools available from HEIDENHAIN, such as **TNCremo** with version **v3.3** or higher, provide all functions for setting up, building up, 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.



Once the connection configurations created in **TNCremo** have been set up, they can be shared among all PC tools for building up 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.



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.

Proceed as follows to set up a secure connection for the logged-on user:

- ▶ Select the **HEROS** menu item
- ▶ Select the **Settings** menu item
- ▶ Select the **Current User** menu item
- ▶ Press the **CERTIFICATES AND KEYS** soft key
- ▶ Select the **Allow password authentication** function
- ▶ Press the **STORE AND RESTART SERVER NOW** soft key
- ▶ Use the **TNCremo** application to set up the secure connection (TCP secure).



For detailed information on how to proceed, please 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, the **Allow password authentication** function is to be deactivated 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. However, this is not described in detail in this document.

Proceed as follows to delete a key in the control so that a user can no longer use the secure DNC connection:

- ▶ Select the **HEROS** menu item
- ▶ Select the **Settings** menu item
- ▶ Select the **Current User** menu item
- ▶ Press the **CERTIFICATES AND KEYS** soft key
- ▶ Select the key to be deleted
- ▶ Press the **DELETE SSH KEY** soft key
- > The control deletes the selected key.

Disabling insecure connections in the firewall

The DNC protocols LSV2 and RPC can be disabled in the firewall to ensure that the control's IT security is really improved through the use of SSH tunnels.

This requires that the following parties change to secure connections:

- Machine tool builders with all additional 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.

- End users with existing DNC connections

If you have ensured that the connections of all parties are secure, the DNC protocol can be disabled in the firewall.

Proceed as follows to disable the DNC protocol in the firewall:

- ▶ Select the **HEROS** menu item
- ▶ Select the **Settings** menu item
- ▶ Select the **Firewall** menu item
- ▶ Select the **Prohibit all** method for **LSV2**
- ▶ Press the **Apply** button
- > The control saves your changes.
- ▶ Close the window with **OK**

Login to user administration

The login dialog appears in the following situations:

- If user administration is active, the dialog appears immediately after the control has booted
- 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

The login dialog provides the following options:

- Users who logged in at least once
- **Other** user

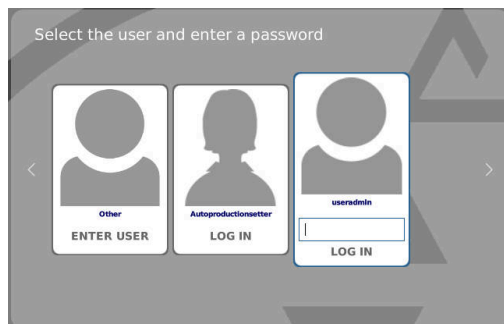
Proceed as follows to log in a user who is already shown in the login dialog:

- ▶ Select the 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.

To log in a user for the first time, you need to use **Other**.

Proceed as follows to log in a user for the first time with **Other**:

- ▶ 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 recognizes the user.
- > The control opens a window with the message **Password expired. Change your password now.**
- ▶ Enter your 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.



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.

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

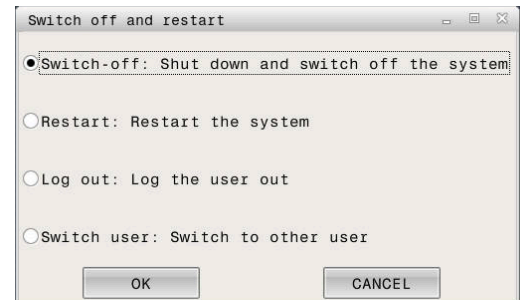


To continue, you need to log in a new user and enter his password.
The NC program continues running under the previously logged-on user.

- **Switch user:**
 - The login window appears
 - The user is not logged out



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.



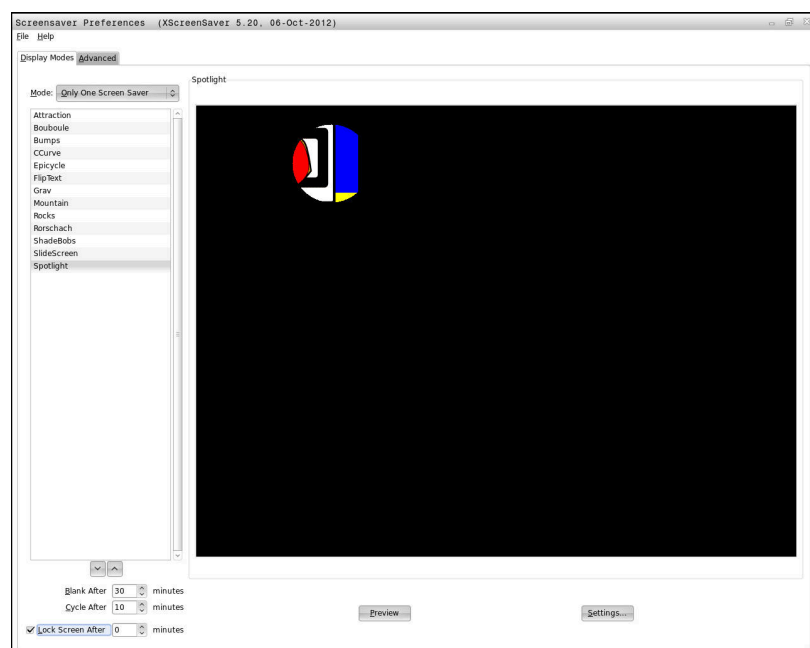
Screensaver with lockout

You can use the screensaver to lock the control. NC programs that have already been started continue running during this period.



You need to enter a password to unlock the screensaver again.

Further information: "Login to user administration", Page 314



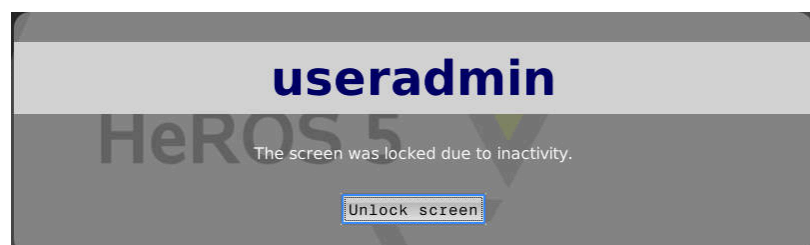
Proceed as follows to access the screensaver settings in the HEROS menu:

- ▶ Select the **HEROS** menu item
- ▶ 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 (e.g. the mouse), the screensaver disappears and the lock screen appears.



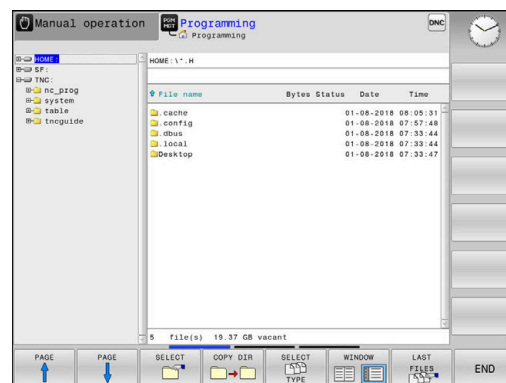
Use **Unlock screen** or Enter to call the login window again.

Further information: "Login to user administration", Page 314

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 logged-on user.



Current User

With **Current User**, you can view the group rights of the currently logged-on user in the **HEROS** menu.



In legacy mode, the **user** function user is automatically logged on to the system during control start-up. When user administration is active, the **user** function user has no effect.

Further information: "HEIDENHAIN function users", Page 304

Calling the **Current User**:

- ▶ Select the **HEROS** menu icon
- ▶ Select the **Settings** menu icon
- ▶ Select the **Current User** menu icon

The user administration allows you to temporarily increase the current user's rights by adding the rights of a user you selected.

Proceed as follows to temporarily increase a user's rights:

- ▶ Call **Current User**
- ▶ Press the **Add rights** soft key
- ▶ Select a user
- ▶ Enter the selected 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 entered in **Add rights**.



By entering code numbers or passwords that replace code numbers, you can temporarily enable rights of **oem** function users.

Further information: "HEIDENHAIN function users", Page 304

You have the following options to remove the rights that have temporarily been added:

- Enter the code number **0**
- Log out the user
- Press the **REMOVE ADDED RIGHTS** soft key

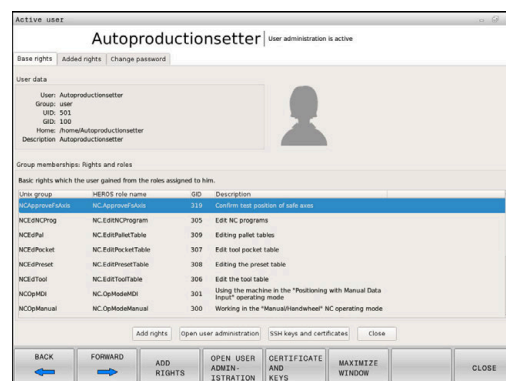
Proceed as follows to select the **REMOVE ADDED RIGHTS** soft key:

- ▶ Call **Current User**
- ▶ Select the **Added rights** tab
- ▶ Press the **REMOVE ADDED RIGHTS** soft key

In the **Current User** menu item, you can change your current user's password.

Proceed as follows to change your current user's password:

- ▶ Call **Current User**
- ▶ Select the **Change password** tab
- ▶ Enter your old password
- ▶ Press the **VERIFY OLD PASSWORD** soft key



- > The control checks whether you have correctly entered your old password.
- > If the control verified that the password is correct, the fields **New password** and **Repeat password** are enabled.
- ▶ Enter your new password
- ▶ Repeat your 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 314

- > The message **Password was changed successfully** appears.

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:

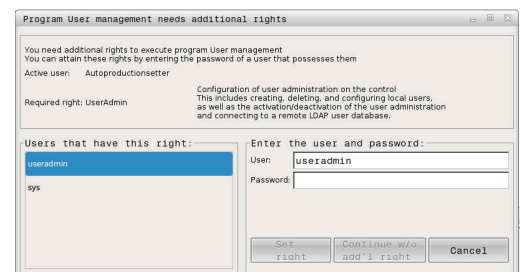
In this window, the control offers you to temporarily increase your rights by adding another user's rights.

In the **Users that have this right:** field, the control lists all existing users that have the right to this function.



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.



Proceed as follows to temporarily add another user's rights to your user's rights:

- ▶ Select a user that possesses the required right
- ▶ 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 318

9.7 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 is adapted to the NC conversational language every time the control is restarted.



In order to change the HEROS conversational language, the **Allow NC to change HEROS config files** function must be activated in the **SELinux** menu.

Further information: "SELinux security software", Page 263

The following link provides information on how to change the NC conversational language:

Further information: "List of user parameter", Page 325

You can change the language layout of the keyboard for HEROS applications.



The language layout for the control and the HEIDENHAIN keyboard always remains in English, even after the language has been changed. Changing the language layout is therefore only reasonable for additional keyboards.

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**
- ▶ Select **OK**
- ▶ Press **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 tool builder can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.

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:



- ▶ Press the **MOD** key










- ▶ If required, use the arrow keys to navigate to **Code-number entry**






- ▶ Enter the code number **123**
- ▶ 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 type of the configuration object is identified by its folder symbol:

-  Key (group name)
-  List
-  Entity (parameter object)



The icons of not yet active parameters and objects 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 **STORE** soft key to save the changes



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.

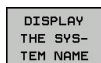
Changing the display of the parameters

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.

Proceed as follows in order to have the actual system names of the parameters be shown:



► 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 parameter

Parameter settings

DisplaySettings

Display sequence and rules for axes

[0] to [5]

Depends on the available axes

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

Sequence of the axes shown in the REF display

[0] to [5]

Depends on the available axes

Type of position display in the position window

NOML

ACTL

REF ACTL

REF NOML

LAG

ACTDST

REFDST

M 118

Type of position display in the status display

NOML

ACTL

REF ACTL

REF NOML

LAG

ACTDST

REFDST

M 118

Definition of decimal separator for position display

. point

Parameter settings

, comma

Feed rate display in Manual Operation mode

at axis key: Display the feed rate only if axis-direction key is pressed

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 control loop

during closed loop and M5: Display spindle position if spindle is in position control loop and with M5

Lock 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 [9]

Depends on the activated options

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

Display step for position display in inches

0.005

0.001

0.0005

0.0001

DisplaySettings

Definition of the unit of measure valid for the display

metric: Use the metric system

inch: Use the inch system

Parameter settings

DisplaySettings

Format of the NC programs and cycle display

Program entry in HEIDENHAIN Klartext conversational programming language or in DIN/ISO

HEIDENHAIN: Klartext conversational programming language for entering the program in Positioning with Manual Data Input operating mode

ISO: DIN/ISO for entering the program in Positioning with Manual Data Input operating mode

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

PLC conversational language

See NC conversational language

PLC error message language

See NC conversational language

Language for online help

See NC conversational language

Parameter settings

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

DisplaySettings

Display mode for time of day

Selection of a display mode for the time of day

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

OFF: Switch off the information line in the operating-mode line.

ON: Switch on the information line in the operating-mode line

DisplaySettings

Settings for 3-D view

Model type of the 3-D view

3-D (compute-intensive): Model depiction for complex machining operations with undercuts

2.5-D: Model depiction for 3-axis operations

No Model: Model depiction is deactivated

Model quality of 3-D view

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

Parameter settings

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

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 and 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

X_Positive, Y_Positive, X_Negative, Y_Negative, Z_Positive, Z_Negative
(depending on the tool axis)

Distance from lower edge of tool to upper edge of stylus

0.001 to 99.9999 [mm]: Offset of stylus to tool

Rapid traverse in probing cycle

10 to 300 000 [mm/min]: Rapid traverse in probing cycle

Probing feed rate during tool measurement

1 to 3000 [mm/min]: Probing feed rate during tool measurement

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 speed

MinSpindleSpeed: Use the minimum spindle speed

Max. permissible surface cutting speed at the tooth edge

1 to 129 [m/min]: Permissible surface cutting speed at the circumference of the milling tool

Maximum permissible speed for tool measurement

0 to 1000 [1/min]: Maximum permissible speed

Maximum permissible measurement error during tool measurement

0.001 to 0.999 [mm]: First maximum permissible measurement error

Maximum permissible measurement error during tool measurement

0.001 to 0.999 [mm]: Second maximum permissible measurement error

Parameter settings

NC stop during tool check

True: When the breakage tolerance is exceeded, the NC program is stopped

False: The NC program is not stopped

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

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

[1]: Y coordinate of the stylus center with respect to the machine datum

[2]: Z coordinate of the stylus center with respect to the machine datum

Set-up clearance above the stylus for pre-positioning

0.001 to 99 999.9999 [mm]: Set-up clearance in tool-axis direction

Safety zone around the stylus for prepositioning

0.001 to 99 999.9999 [mm]: Set-up clearance in the plane perpendicular to the tool axis

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 tool builder

Configuration of the fixed cycles

Overlap factor for pocket milling

0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING

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**

M function for spindle orientation in fixed cycles

-1: Spindle orientation directly by NC**0: Function inactive****1 to 999: Number of the M function for spindle orientation**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**

Parameter settings

Do not show the **Residual material** warning

On: Warning is not displayed

Off: Warning is displayed

Special spindle parameters for threads

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

–999999999 to 999999999: The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation

Advanced switching time of spindle

–999999999 to 999999999: The spindle is stopped at this time before reaching the bottom of the thread

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.

100 to 50000: Program length for which the geometry is to be checked

DIN/ISO programming: Block number increment

0 to 250: Increments for creation of DIN/ISO blocks in the program

Specify programmable axes

TRUE: Use the specified axis configuration

FALSE: Use the default axis configuration XYZABCUVW

Line number to which identical syntax elements are searched for

500 to 50000: Search for selected elements with up or down arrow keys

Behavior of PARAXMODE function with UVW axes

FALSE: PARAXMODE function enabled

TRUE: PARAXMODE function disabled

Parameter settings

File management settings

Display of dependent files

MANUAL: Dependent files are displayed

AUTOMATIC: Dependent files are not displayed

Settings for tool-usage files

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 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

These machine parameters are only effective for a Windows programming station

List of drives and/or directories

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

Serial Interface RS232

Further information: "Setting up data interfaces", Page 276

10.2 Pin layout and cables for data interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices



The interface complies with the requirements of EN 50178 for **Low voltage electrical separation**.

When using the 25-pin adapter block:

Control		Conn. cable 365725-xx			Adapter block 310085-01		Conn. cable 274545-xx		
Male	Assign-ment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	White/ Brown	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

When using the 9-pin adapter block:

Control		Conn. cable 355484-xx		Adapter block 363987-02		Conn. cable 366964-xx			
Male	Assign-ment	Female	Color	Male	Female	Male	Female	Color	Female
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	White/ Green	8	8	8	8	White/ Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	External shield	Hsg.	External shield	Hsg.	Hsg.	Hsg.	Hsg.	External shield	Hsg.

Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from that of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363987-02		Conn. cable 366964-xx		
Female	Male	Female	Color	Female
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	External shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

10.3 Specifications

Explanation of symbols

- Default
- Axis option
- 1 Advanced Function Set 1

Specifications

Components	<ul style="list-style-type: none"> ■ Operating panel ■ Screen with soft keys
Program memory	<ul style="list-style-type: none"> ■ 2 GB
Input resolution and display step	<ul style="list-style-type: none"> ■ As fine as 0.1 µm for linear axes ■ Up to 0.0001° for rotary axes
Input range	<ul style="list-style-type: none"> ■ Maximum 999 999 999 mm or 999 999 999°
Block processing time	<ul style="list-style-type: none"> ■ 6 ms
Axis feedback control	<ul style="list-style-type: none"> ■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 3 ms ■ Cycle time of speed controller: 200 µs
Range of traverse	<ul style="list-style-type: none"> ■ Maximum 100 m (3937 inches)
Spindle speed	<ul style="list-style-type: none"> ■ Maximum 100,000 rpm (analog speed command signal)
Error compensation	<ul style="list-style-type: none"> ■ Linear and nonlinear axis error, backlash, thermal expansion ■ Static friction
Data interfaces	<ul style="list-style-type: none"> ■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the control through the data interface with the TNCremo software ■ Ethernet interface 1000 BaseT ■ 3 x USB (1 x front USB 2.0; 2 x rear USB 3.0)
Ambient temperature	<ul style="list-style-type: none"> ■ Operation: 5 °C to +45 °C ■ Storage: -35 °C to +65 °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 with TOOL CALL . 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	–9.9999 to +9.9999 (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 999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–99 999.9999 to +99 999.9999 (9, 6)
Labels (LBL) for program jumps	0 to 999 (5, 0)
Labels (LBL) for program jumps	Any text string in quotation marks (""")
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

Short description	<ul style="list-style-type: none"> ■ Basic version: 3 axes plus closed-loop spindle □ 1. Additional axis for 4 axes plus closed-loop spindle □ 2. Additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational format
Position entry	<ul style="list-style-type: none"> ■ 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
Parallel operation	Creating an NC program with graphical support while another NC program is being run
Cutting data	Automatic calculation of spindle speed, cutting speed, feed per tooth and feed per revolution
Program jumps	<ul style="list-style-type: none"> ■ Subprograms ■ Program section repeats ■ Any desired NC program as subprogram
Machining cycles	<ul style="list-style-type: none"> ■ Cycles for drilling, and conventional and rigid tapping ■ Roughing and finishing rectangular pockets ■ Cycles for pecking, reaming, boring, and counterboring ■ 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	<ul style="list-style-type: none"> ■ Datum shift, mirroring ■ Scaling factor (axis-specific)
Q parameters Programming with variables	<ul style="list-style-type: none"> ■ Mathematical functions =, +, -, *, /, roots ■ Logical operations (=, ≠, <, >) ■ Calculating with parentheses ■ sin α, cos α, tan α, arc sin, arc cos, arc tan, aⁿ, eⁿ, 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 param.
Programming aids	<ul style="list-style-type: none"> ■ Calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ TNCguide: The integrated help system. ■ Graphic support for the programming of cycles ■ Comment and structure blocks in the NC program

User functions

Teach-In	<ul style="list-style-type: none"> ■ Actual positions can be transferred directly to the NC program
Test graphics Display modes	<ul style="list-style-type: none"> ■ Graphic simulation before a program run, even while another NC program is being run ■ Plan view / projection in 3 planes / 3-D view ■ Detail enlargement
Programming graphics	<ul style="list-style-type: none"> ■ In the Programming operating mode, the contours of the NC blocks are drawn on screen while they are being entered (2-D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	<ul style="list-style-type: none"> ■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ For saving any presets
Contour, returning to	<ul style="list-style-type: none"> ■ 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	<ul style="list-style-type: none"> ■ Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	<ul style="list-style-type: none"> ■ Calibrating the touch probe ■ Presetting, manual ■ Tools can be measured automatically

Accessories

Accessories

Electronic handwheels	■	HR 510: Portable handwheel
	■	HR 550FS: Portable radio handwheel with display
	■	HR 520: Portable handwheel with display
	■	HR 420: Portable handwheel with display
	■	HR 130: Panel-mounted handwheel
	■	HR 150: Up to three panel-mounted handwheels via handwheel adapter HRA 110
<hr/>		
Touch probes	■	TS 248: 3-D touch trigger probe with cable connection
	■	TS 260: 3-D touch trigger probe with cable connection
	■	TT 160: 3-D touch trigger probe for tool measurement
	■	KT 130: Simple touch trigger probe with cable connection

Index

3

- 3-D touch probe
 - Calibrating..... 168
 - using..... 161

A

- About this manual..... 22
- Accessories..... 94
- Adding comments..... 198
- Automatic tool measurement... 105
- Axis position, checking..... 131

B

- Backup..... 270
- Behavior after receipt of ETX.... 279
- Block check character..... 278
- Block scan..... 212
 - after a power failure..... 212
 - In a point table..... 216
- Browser..... 76

C

- Calling tool management..... 118
- Code number entry..... 234
- Condition of RTS line..... 278
- Configuration data..... 322
- Context-sensitive help..... 87
- Control panel..... 50
- Counter..... 228
- Counter settings..... 241

D

- Data Backup..... **72**, 270
- Data interface..... 276
 - pin layout..... 336
 - setting up..... 276
- Data transfer
 - File system..... 278
 - Software..... 280
- Data transmission
 - Behavior after receipt of ETX..... 279
 - Block check character..... 278
 - Condition of RTS line..... 278
 - Data bits..... 277
 - Handshake..... 278
 - Parity..... 277
 - Protocol..... 277
 - Software TNC server..... 279
 - Stop bits..... 277
- Datum table
 - transferring probed values.... 166
- Directory..... **62**
- Display handwheel..... 137
- Displaying an HTML file..... 76
- Displaying an Internet file..... 76

- Display of the NC program..... 198
- Display screen..... 49
- DNC..... 274
- Downloading help files..... 92

E

- EnDat encoder..... 131
- Error message..... 82
 - help with..... 82
- Ethernet interface..... 282
 - configuration..... 283
 - configuring..... 289
 - connecting and disconnecting a network drive..... 70
 - connection possibility..... 282
 - introduction..... 282
- External access..... 245
- External data transfer..... 69

F

- FCL..... 234
- FCL function..... 26
- Feature Content Level..... 26
- Feed rate..... 148
 - changing..... 149
- File
 - importing..... 72
 - protecting..... 64
- File management..... **60**
 - external data transfer..... 69
 - External file types..... 62
- File manager
 - Calling..... 63
 - Directory..... 62
 - File type..... 60
 - Selecting files..... 65
- File status..... 63
- Firewall..... 273
- FUNCTION COUNT..... 228
- Fundamentals..... 93

G

- GOTO..... 197
- Graphics..... 182
 - view options..... 183
- Graphic settings..... 240
- Graphic simulation..... 188
 - tool..... 184

H

- Handwheel..... 136
- Hard disk..... 60
- Help system..... 87
- Help with error message..... 82

I

- Import
 - file from iTNC 530..... 72
 - table from iTNC 530..... 108

- Indexed tool..... 100
- Interrupt machining..... 204
- iTNC 530..... 46

J

- Jumping
 - with GOTO..... 197

K

- Kinematics..... 242

L

- Loading the machine configuration. 236

M

- M91, M92..... 225
- Machine parameters..... 322
 - changing..... 322
 - changing the display..... 324
 - list..... 325
- Machine settings..... 242
- Manual datum setting
 - circle center as preset..... 175
- Manual datum setting..... 174
- Manual presetting
 - On any axis..... 174
 - Setting a center line as preset..... 178
 - without a 3-D touch probe... 158
- MDI..... 219
- Measuring workpieces..... 179
- Miscellaneous functions..... 223
 - enter..... 223
 - For program run inspection. 224
 - For spindle and coolant..... 224
- Miscellaneous functions for coordinate entries..... 225
- Modes of Operation..... 52
- MOD function..... 232
 - exiting..... 232
 - overview..... 233
 - selecting..... 232
- Move machine axes
 - Jog positioning..... 135
- Move the machine axes
 - with the display handwheel. 137
- Moving the machine axes..... 134
 - With axis direction keys..... 134
 - With the handwheel..... 136

N

- NC error message..... 82
- NC program
 - structuring..... 201
- Network connection..... 70
- Network settings
 - control-specific..... 289
 - general..... 283

O

Opening a BMP file.....	80
Opening a GIF file.....	80
Opening a graphics file.....	80
Opening a JPG file.....	80
Opening an INI file.....	79
Opening a PNG file.....	80
Opening a TXT file.....	79
Opening a video file.....	80
Opening Excel files.....	75
Operating times.....	251

P

Path.....	62
PDF viewer.....	74
Pin layout	
data interface.....	336
Pocket table.....	110
Positioning.....	219
with manual data input.....	219
Preset	
managing.....	151
Preset table.....	151, 151
transferring probed values....	167
Probing	
with a 3-D touch probe.....	161
With end mill.....	159
Program	
structuring.....	201
Program run.....	199
block scan.....	212
Interrupt.....	204
measuring.....	190
Overview.....	199
Resuming after interruption.	208
Retraction.....	209
running.....	200
skipping NC blocks.....	192
Program Test	
Overview.....	194
Protection zone.....	243

Q

Q parameters	
checking.....	202

R

Radio handwheel	
configuring.....	248
Restore.....	270
Retraction.....	209
After a power interruption....	209
Returning to the contour.....	217
Rotating, zooming and moving a graphic.....	186

S

Save service files.....	86
-------------------------	----

Screen keypad.....	50, 51
Screen layout.....	49
Set BAUD rate.....	276
Set data transmission speed....	276
Shifting the sectional plane.....	188
Software number.....	234
Spindle speed	
changing.....	149
Status display.....	54
additional.....	55
General.....	54
Stop at.....	196
Structuring NC programs.....	201
Switch-off.....	133
Switch-on.....	130
System settings.....	251

T

Taskbar.....	255
test run.....	218
executing up to a certain NC	
block.....	196
Execution.....	195
Setting speed.....	187
Text file	
opening.....	79
TNCguide.....	87
TNCremo.....	280
Tool carrier management.....	123
Tool change.....	113
Tool data.....	98
entering into the table.....	103
indexing.....	107
Tool length.....	98
Tool management.....	117
Editing.....	119
Tool measurement.....	105
Tool name.....	98
Tool number.....	98
Tool radius.....	98
Tool table.....	99
Edit, exit.....	106
Editing functions.....	106
filter function.....	101
fundamentals.....	99
importing.....	108
input options.....	103
Tool usage file.....	114, 244
Tool usage test.....	114
Touch probe cycles.....	161
manual.....	161
Manual operation mode.....	161
Traverse limits.....	243
Traversing reference marks.....	130

U

USB device	
Connecting.....	68

Removing.....	69
User administration.....	292
User parameters.....	322, 325
Using touch probe functions with mechanical probes or measuring dials.....	160

V

Version number.....	234
Version numbers.....	236

W

Window Manager.....	254
Wireless handwheel.....	140
Assign handwheel holder....	248
Selecting transmitter power.	249
Setting channel.....	249
Statistical data.....	250
Working space monitoring.....	189
Workspace monitoring.....	195
Writing probing values	
log.....	166
to the datum table.....	166
to the preset table.....	167

Z

ZIP archive.....	78
------------------	----

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

E-mail: service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

E-mail: service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106

E-mail: service.app@heidenhain.de

www.heidenhain.de

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

The Klartext on Your
Mobile Device

Google
Play Store

Apple
App Store



Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the
dimensional accuracy of the finished workpieces.

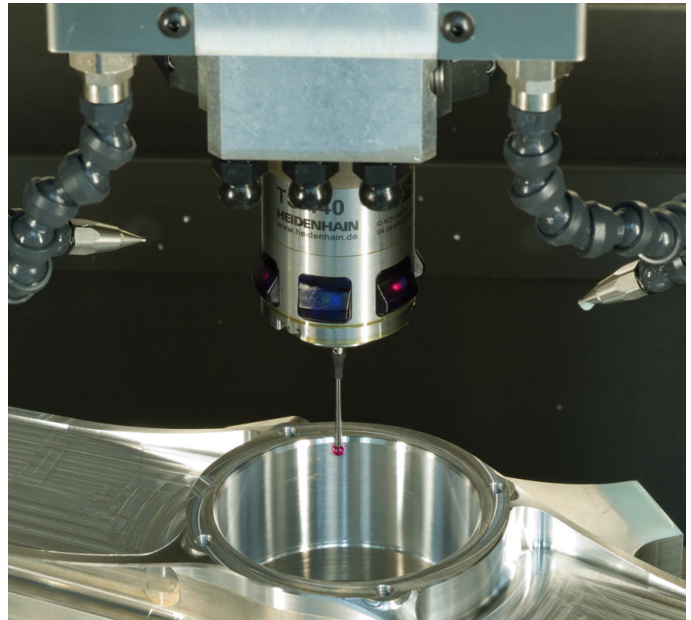
Workpiece touch probes

TS 220 Signal transmission by cable

TS 440, TS 444 Infrared transmission

TS 640, TS 740 Infrared transmission

- Workpiece alignment
- Setting presets
- Workpiece measurement



Tool touch probes

TT 140 Signal transmission by cable

TT 449 Infrared transmission

TL Non-contacting laser systems

- Tool measurement
- wear monitoring
- tool breakage detection

